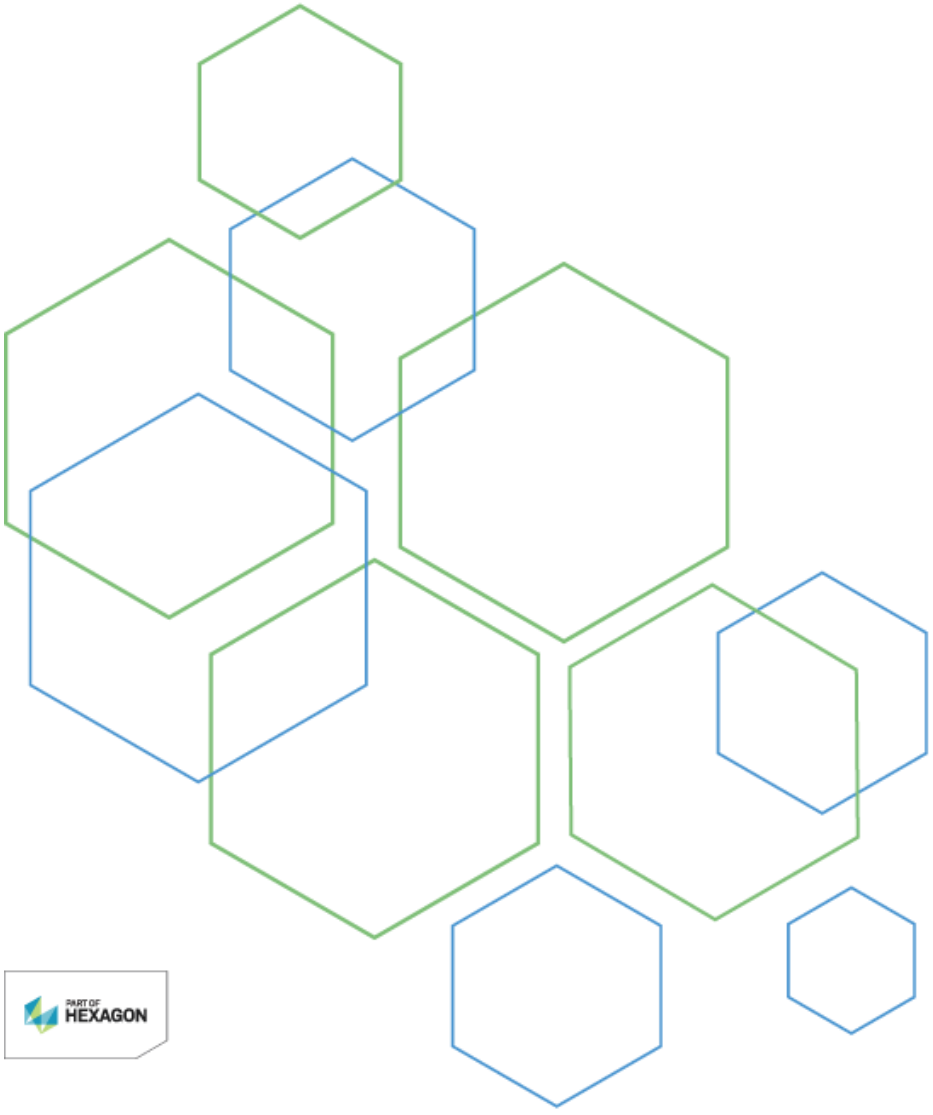
INTERGRAPH Smart 3D

Planning

User's Guide



PART OF HEXAGON

Version 2016 (11.0)
November 2016

Copyright

Copyright © 1999-2016 Intergraph® Corporation. All Rights Reserved. Intergraph is part of **Hexagon**.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2016. All Rights Reserved.

Portions of the user interface are copyright © 2012-2016 Telerik AD.

U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial use. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.

b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, http://www.bis.doc.gov/complianceandenforcement/liststocheck.htm, the U.S. Department of Treasury Specially Designated Nationals List, http://www.treas.gov/offices/enforcement/ofac/, and the U.S. Department of State Debarred List, http://www.pmddtc.state.gov/compliance/debar.html.

c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.

d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Intergraph, the Intergraph logo, PDS, SmartPlant, FrameWorks, I-Sketch, SmartMarine, IntelliShip, ISOGEN, SmartSketch, SPOOLGEN, SupportManager, SupportModeler, Sapphire, and Intergraph Smart are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Hexagon and the Hexagon logo are registered trademarks of Hexagon AB or its subsidiaries. Microsoft and Windows are registered trademarks of Microsoft Corporation. ACIS is a registered trademark of SPATIAL TECHNOLOGY, INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, ProtoViewCtl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Incorporates portions of 2D DCM, 3D DCM, and HLM by Siemens Product Lifecycle Management Software III (GB) Ltd. All rights reserved. Gigasoft is a registered trademark, and ProEssentials a trademark of Gigasoft, Inc. VideoSoft and VXFlexGrid are either registered trademarks or trademarks of ComponentOne LLC 1991-2013, All rights reserved. Oracle, JD Edwards, PeopleSoft, and Retek are registered trademarks of Oracle Corporation and/or its affiliates. Tribon is a trademark of AVEVA Group plc. Alma and act/cut are trademarks of the Alma company. Other brands and product names are trademarks of their respective owners.

Contents

Preface **8**

What's New in Planning **8**

Planning **10**

Assembly Hierarchy 14
Icons in the Workspace Explorer 17
Plane Methods 24
Define a coincident plane 25
Define an offset plane 26
Define plane using angle from plane 27
Define a plane using a point and a normal vector 28
Define a plane using three points 29
Define plane by vectors normal 29
Editing Properties 30
Common Property Tabs 32
Permission Group Behaviors between Tasks 35
Split Notification 39
Add a seam 40
Delete a seam 41
Modify a seam when leaf systems are not added or removed 42

Split Block **43**

Split a block 49
Modify a block 49
Modify the size of the top-level block (B0) 50
Set block manufacturing orientation by baseplate 50
Set block manufacturing orientation by three-point plane 51
Block Properties Dialog Box 51
General Tab (Block Properties Dialog Box) 52
Select or Modify Frames Dialog Box 56

Manage Block Intersections **58**

Manage intersections 61
Use a drag-and-drop operation to assign parts 62
Manage Block Intersections Dialog Box 62

New (Block) Assembly **64**

Create a new assembly or assembly block 69
Set assembly user-defined planar manufacturing orientation 70
Set assembly manufacturing orientation by object 71
Edit an assembly or assembly block 72
Assembly Properties Dialog Box 72

General Tab (Assembly Properties Dialog Box) 72

Merge 77

Merge blocks or assemblies 78

Sequence 79

Define the assembly sequence 80
Assembly Sequence Form Dialog Box 80

Mirror Assembly 83

Mirror the assembly hierarchy 85

Auto Assembly 87

Create a rule-based assembly 88
SMAutoAssemblyRule 88
BuiltUpAutoAssemblyRule 89

Generate Planning Joints 90

Generate a planning joint 95
Edit a planning joint 95
Create a new weld group folder 96
Move a planning joint to a different group folder 96
Move a planning joint to a different assembly 97
Schedule a planning joint using Batch Services 97
Planning Joint Properties Dialog Box 97
General Tab (Planning Joint Properties Dialog Box) 98

Split Planning Joint 106

Split a planning joint 107

Spooling 108

Generate Spools 110
Create spools 110
Edit spool properties 111
Delete a spool 111
Spool Generation Dialog Box 112
Spool Properties Dialog Box 116
Create Penetration Spools 117
Create penetration spools 117
Penetration Spool Generation Dialog Box 118
Penetration Spool Properties Dialog Box 120

Generate Production Routing 127

Apply rule for production routing 129

Create Common Part Managers **130**

Create common part managers 131

Common Part Group **132**

Create a common part group 133
Select a common part group using filters 133
Common Part Group Properties Dialog Box 133
General Tab 134

Manufacturing Service Manager **135**

Review a manufacturing object 137
Update a manufacturing object 137
Review or update a set of objects selected by a filter 137
Review Dialog Box 138
Selection of Possible Report Data Dialog Box 139

Submit Batch Job **140**

Run a detailing batch process 141
Run a production batch process 141
Run a custom batch process 142
Add a custom batch process 142
Generic Batch Command Dialog Box 143
Batch Process Selection Dialog Box 144

Un-Split **145**

Un-split a plate at a selected seam 145
Seam Properties Dialog Box 145
Main Tab (Seam Properties Dialog Box) 146

Run Report **147**

Assembly Views **148**

Show Selected 148
Show selected objects 148
Show Parts 148
Show parts 149
Show Assembly View 149
Show the assembly view 150
Show Assembly Sequence 150
View the assembly sequence 153

Assembly Arrangements **154**

Synchronize Hierarchy 154
Synchronize the assembly hierarchy 155
Block Assignment 156
Start the block assignment process 158
Stop the block assignment process 159

Schedule block assignment using Batch Services 159
Create block assignment log file 160
Schedule [Task] Dialog Box 160
Optional Schedule Properties Dialog Box 161
Import Assembly Data from Excel 161
Import Assembly Data from Excel Dialog Box 162

Common Parts 163

Common Part Service 163
Start Common Part Service 165
Common Part Explorer 166
View Common Parts 167
Remove Common Parts 167
Modify Common Part Groups 167
Reevaluate parts for common part managers 168
Update a common part manager 168
Update a common part folder 168
Common Part Manager Properties Dialog Box 168
Common Part Folder Properties Dialog Box 169
Common Part Utilities 169
Test Rules 169
Compare an object type 170
Compare an XML 171
Edit Rule Settings Dialog Box 171

Planning Utilities 197

Repair Planning Objects 197
Repair Planning objects 197
Repair Planning Objects Dialog Box 197
Test Block Assignment 200
Test block assignment 200
Test Block Assignment Dialog Box 200

Check Manufacturability 202

Check objects for manufacturability 203
Schedule check manufacturability using Batch Services 203
Manufacturability Checking Settings Dialog Box 204
Show Manufacturability Inconsistencies Dialog Box 205

Flatten Block/Assembly/Assembly Block 210

Flatten an assembly or block 210

Glossary 211

Index 229

Preface

This document is a user's guide for the Planning functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (*https://smartsupport.intergraph.com*).

What's New in Planning

The following changes have been made to the Planning task:

Version 2016 (11.0)

- The **Import Assembly Data from Excel** command creates or deletes assembly hierarchies or assigns parts to an assembly hierarchy from a Microsoft Excel spreadsheet. For more information, see *Import Assembly Data from Excel* (on page 161). (P2 CP:181796)
- Added the **Submit Batch Job** command to run batch processes using the Intergraph Batch Services framework. For more information, see *Submit Batch Job* (on page 140). (P2 CP:273093, P2 CP:178040)
- A new locate filter, **Construction Graphics** is available. For more information, see *Editing Properties* (on page 30). (P2 CP:271166)
- The **Sequence** tab is removed from the **Block Properties** and **Assembly Properties** dialog boxes. A new command, **Sequence Form**, allows you to create, review, and modify the assembly sequence. For more information, see *Sequence* (on page 79). (P2 CP:281691)
- Updated the **Weld Inclination**, **Weld Rotation**, and **Weld Position** descriptions on the **Planning Joint Properties** dialog box. For more information, see *General Tab (Planning Joint Properties Dialog Box)* (on page 98). (P2 CP:245039 and P2 CP:285193)
- Updated the **Common Part** rule settings documentation to better explain the Check Display Text options. For more information, see *Comparison Tab (Edit Rule Settings Dialog Box)* (on page 172). (P2 CP:288158)
- The **Generate Planning Joints** command contains new actions and rules. Additional properties are available for planning joints. For more information, see *Generate Planning Joints* (on page 90) and *General Tab (Planning Joint Properties Dialog Box)* (on page 98). (P3 CP:261092)
- A new option, **Task List**, has been added to the **Manufacturability Checking Settings** dialog box. For more information, see *Check Manufacturability* (on page 202). (P3 CP:288433)
- Added properties for planning joints. For more information, see *General Tab (Planning Joint Properties Dialog Box)* (on page 98). (P3 CP:279461)
- You can define base plane rotation and update relationships from the **New (Block) Assembly** command ribbon. For more information, see *New (Block) Assembly* (on page 64). (P3 CP:280175)

- You can drag planning joints to different planning joint group folders and assemblies. Custom folders are also available. For more information, see *Generate Planning Joints* (on page 90). (P3 CP:280175)
- The **Show Assembly Sequence** ribbon contains new commands for displaying weight, range, and assembly sequence properties. For more information, see *Show Assembly Sequence* (on page 150). (P3 CP:289130)
- Two new commands are available on the **Tools** menu: **Repair Planning Objects** and **Test Block Assignment**. For more information, see *Planning Utilities* (on page 197). (P3 CP:173117)
- Updated the **Show Assembly Sequence** command documentation to reflect the changes in the interface. For more information, see *Show Assembly Sequence* (on page 150). (P4 CP:273773)

SECTION 1

Planning

The Planning task lets you perform production planning and define the assembly hierarchy of the model structure by assigning light and detailed parts to blocks and assemblies. The assembly hierarchy, after attributes are assigned, is analogous to a production bill of material.

The software creates a default initial block enclosing all structure in the model when the Planning task is first used. This block is called **B0** and is at the top level of the hierarchy. You create an assembly hierarchy containing the following:

- **Block** - High-level assembly that is a spatial definition of a section of the marine structure. Blocks are created by splitting a volume and contain assemblies, parts, and/or child blocks created by splitting the block. A block is a child of the top-level block (B0), an assembly block, or another block.
- **Assembly** - Collection of assemblies, structural parts, equipment, outfitting parts, and outfitting assemblies (such as pipe spools). An assembly is a child of a block, assembly block, or another assembly.
- **Assembly Block** - Collection of blocks, assemblies, structural parts, equipment, outfitting parts, and outfitting assemblies (such as pipe spools). An assembly block is a child of the top-level block or another assembly block.

You define blocks by splitting and merging existing blocks. New blocks automatically contain all parts that are entirely within the boundary of the block. All parts that are partially within the boundaries are assigned by the software to a higher-level block that fully contains the parts. Parts that are automatically assigned to blocks are put in a folder called **UnAssigned Parts**. When you have manually reassigned all the parts to a subassembly or a neighboring block, the **UnAssigned Parts** folder no longer displays.

You must either set up the block assignment process to run continuously on a server, or activate it periodically for the automatic assignment of parts to blocks to work. For more information about the block assignment process, see *Block Assignment* (on page 156).

With the **Manage Block Intersections**  command, you can control the parts that intersect block boundaries. You can reassign parts to any of the intersecting blocks or split intersected parts using the dialog box.

You define assemblies and assembly blocks by selecting the child objects, and then selecting the parent assembly, assembly block, or block.

You can drag objects to any block or assembly in the **Workspace Explorer**, even if the part is outside of the boundaries of a block.

You can assign properties such as Workcenter, Production Equipment, Assembly Stage, and Type to blocks and assemblies. You define the allowed properties in the Planning reference data. You can also define the manufacturing orientation of an assembly or block.

The assembly sequence lets you define the building sequence of blocks and assemblies in a workcenter or the dock. You can also manipulate the model view using commands in both the Planning and Common tasks.

You start the Planning task by clicking **Tasks > Planning**. The Planning task has the following task-specific commands located on the vertical toolbar:

	**Select** - Selects objects in the model. For more information, see *Editing Properties* (on page 30).
	**Split Block** - Designs new blocks by splitting existing blocks with the cutting plane. For more information, see *Split Block* (on page 43).
	**Manage Block Intersections** - Identifies the outfitting and structural parts that intersect with a block cutting plane, and allows you to assign the part to a block or split the part if it is a structural part. For more information, see *Manage Block Intersections* (on page 58).
	**New (Block) Assembly** - Designs assemblies and assembly blocks and define properties for their associated building method. For more information, see *New (Block) Assembly* (on page 64).
	**Merge** - Merges the content of two or more blocks, assemblies, or assembly blocks into one block or assembly. For more information, see *Merge* (on page 77).
	**Sequence** - Defines the details of the sequence in which individual blocks, assemblies, and assembly blocks of the model structure are to be built. For more information, see *Sequence* (on page 79).
	**Mirror Assembly** - Copies a subset of the assembly hierarchy structure from one side of the model structure to the other and automatically place symmetrical parts into the mirrored assembly. For more information, see *Mirror Assembly* (on page 83).
	**Auto Assembly** - Automatically creates assemblies based on pre-defined rules. For more information, see *Auto Assembly* (on page 87).
	**Generate Planning Joints** - Creates planning joints on the physical connections in a block, assembly, or assembly block. A planning joint breaks a physical connection into its individual manufacturing components, such as the individual fillets on a double fillet weld. For more information, see *Generate Planning Joints* (on page 90).
	**Split Planning Joint** - Splits a planning joint if, for example, another weld bevel is needed. For more information, see *Split Planning Joint* (on page 106).
	**Generate Spools** - Combines pipe parts and components (such as flanges and welded valves) into spools. This command is identical to the one found in the Piping task. For more information, see *Generate Spools* (on page 110).
	**Create Penetration Spools** - Creates penetration spools. This special type of spool consists of one or more spools penetrating a plate part. There is a tight connection between the spools and the plate part using a penetration plate created in the Hangers and Supports task. For more information, see *Create Penetration Spools* (on page 117).

	**Generate Production Routing** - Generates production data for curved parts. This command allows you to assign production information such as Workcenter or Stage Code, and assign different routing actions. For more information, see *Generate Production Routing* (on page 127).
	**Create Common Part Managers** - Processes objects and creates common part managers. For more information, see *Create Common Part Managers* (on page 130).
	**Common Part Group** - Creates common part groups. Common part groups are also created as an entity during the **Common Part Service** process. For more information, see *Common Part Group* (on page 132).

The Planning task has the following task-specific commands located on the **Tools** menu or **Workspace Explorer**:

Tools menu	**Manufacturing Service Manager** - Updates or reviews manufacturing objects. For more information, see *Manufacturing Service Manager* (on page 135).
Tools menu	**Submit Batch Job** - Runs batch processes using the Intergraph Batch Services framework. This command is available on the **Tools** menu. For more information, see *Submit Batch Job* (on page 140).
Tools menu	**Un-Split** - Merges plates into a single plate leaving the seam in the model so that you can use the seam to split the plate again at a later time. For more information, see *Un-split a plate at a selected seam* (on page 145).
Tools > Assembly Views menu & Workspace Explorer	**Show Parts** - Allows you to quickly view the parts that are directly assigned to the selected assembly. This command is available on the **Tools > Assembly Views** menu or on the shortcut menu of the **Workspace Explorer**. For more information, see *Show Parts* (on page 148).
Tools > Assembly Views menu	**Show Selected** - Allows you to quickly hide all but the selected objects. This command is available on the **Tools > Assembly Views** menu. For more information, see *Show Selected* (on page 148).
Tools > Assembly Views menu & Workspace Explorer	**Show Assembly View** - Allows you to quickly display an isometric view of a selected block or assembly in its manufacturing orientation. This command is available on the **Tools > Assembly Views** menu or on the shortcut menu of the **Workspace Explorer**. For more information, see *Show Assembly View* (on page 149).
Tools > Assembly Views menu	**Show Assembly Sequence** - Uses assembly sequence information and manufacturing orientation (the orientation, or up-vector, of an assembly while being built) to show a step-by-step animation of the assembly method for a selected block and/or assembly. This command is available on the **Tools > Assembly Views** menu. For more information, see *Show Assembly Sequence* (on page 150).

Tools > Assembly Arrangements menu	**Synchronize Hierarchy** - Performs a bi-directional assignment of symmetrical **UnAssigned Parts** into symmetrically related blocks and assemblies. This command is available on the **Tools > Assembly Arrangements** menu. For more information, see *Synchronize Hierarchy* (on page 154).
Tools > Assembly Arrangements menu	**Block Assignment** - Starts and stops the Planning query process. This command is available on the **Tools > Assembly Arrangements** menu. For more information, see *Block Assignment* (on page 156).
Tools > Assembly Arrangements menu	**Import Assembly Data from Excel** - Imports an assembly hierarchy defined in a Microsoft Excel file. You can also delete assembly hierarchies and assign parts to assembly hierarchies. This command is available on the **Tools > Assembly Arrangements** menu. For more information, see *Import Assembly Data from Excel* (on page 161).
Tools > Common Part menu	**Common Part Service** - Identifies and manages structural parts that are common within a block, assembly, or assembly block. For more information, see *Common Part Service* (on page 163).
Tools > Common Part menu & **Workspace Explorer**	**Common Part Explorer** - Displays the common parts as groups under folders of the relevant block, assembly or assembly block. This command is available on the **Tools** menu or on the shortcut menu of the **Workspace Explorer**. For more information, see *Common Part Explorer* (on page 166).
Tools > Common Part menu	**Common Part Utilities** - Consists of the following common part utilities. ▪ **Test Rules** - Displays the values of the properties of an object type or an XML after comparison. ▪ **Edit/Review Rule Settings** - Displays the rule settings to compare objects of the same type, such as the filters and the rule variables. For more information, see *Common Part Utilities* (on page 169).
Tools menu	**Check Manufacturability** - Analyzes pipes in the current workspace and reports the objects that will be difficult or impossible to manufacture. This command is available on the **Tools** menu. For more information, see *Check Manufacturability* (on page 202).
Workspace Explorer	**Flatten Block/Assembly** - Removes an assembly, block, or assembly block from the assembly hierarchy without deleting its child parts, assemblies, or blocks. This command is only available for the selected block or assembly on the shortcut menu of the **Workspace Explorer**. For more information, see *Flatten Block/Assembly/Assembly Block* (on page 210).
Workspace Explorer	**Assigning parts to blocks or assemblies in the Workspace Explorer** - Allows you to assign any part to any block or assembly by a drag-and-drop operation in the **Workspace Explorer**.

Dual Workspace Explorers

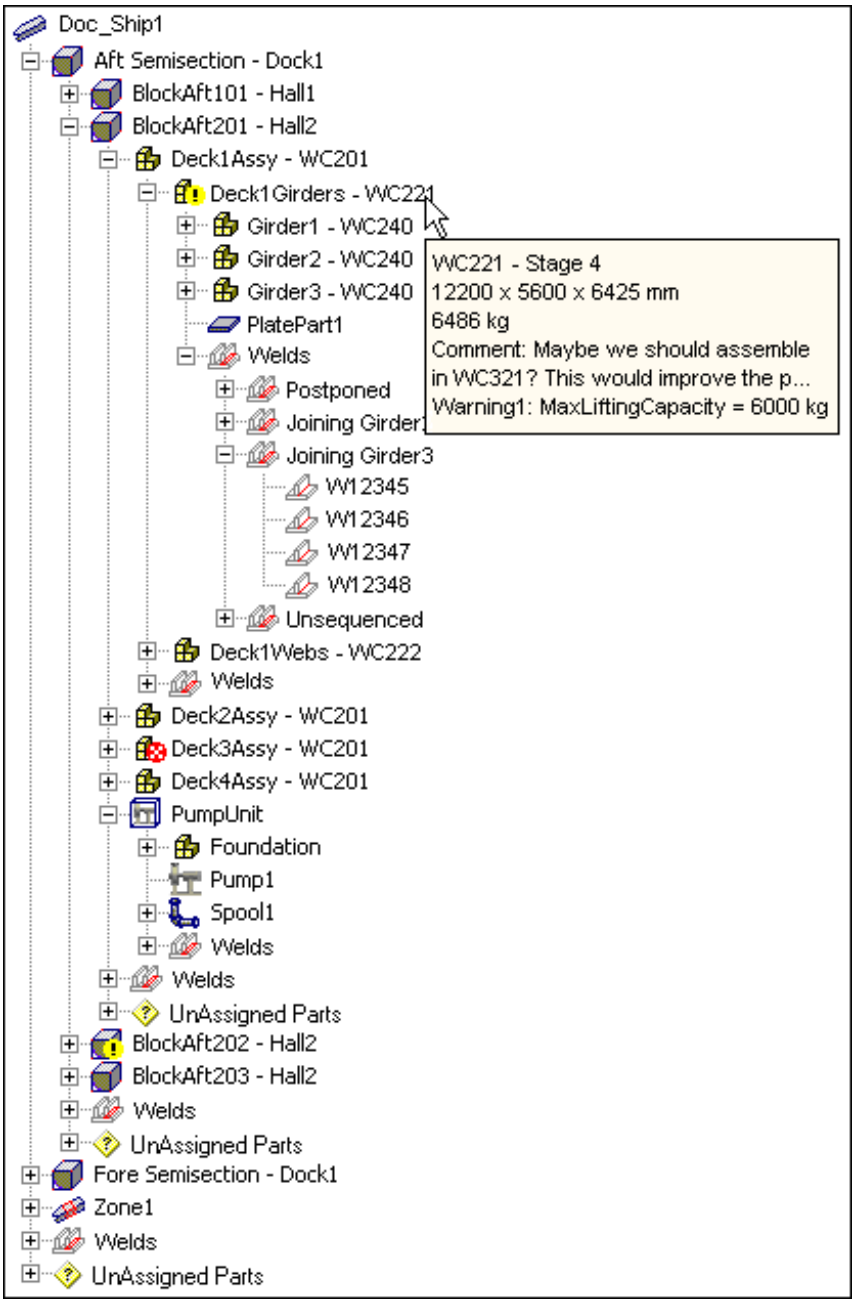
In the Planning task, you can start a second **Workspace Explorer** by clicking **View** > **Workspace Explorer 2**. You can perform the following common operations within and across the **Workspace Explorers**:

- Drag objects to any block or assembly within the same **Workspace Explorer**, the changes are also reflected on the other.
- Drag objects to any block or assembly from one **Workspace Explorer** to the other. You cannot drag objects from the **Assembly** tab of one **Workspace Explorer** to the **System** tab of the other or vice versa.
- **Lock Tree** - Select this option for a **Workspace Explorer** to lock its node expansion.

Assembly Hierarchy

The primary purpose of the Planning task is to define and document *how* and *where* to build model structure and its subassemblies (production planning), rather than *when* to build the structure (production scheduling). The result is an assembly hierarchy analogous to a manufacturing bill of material that can be considered the specification of how to build the

structure (assemblies and their assembly sequence). To view the assembly hierarchy, click the **Assembly** tab in the **Workspace Explorer**.



The assembly hierarchy is attributed with assembly information, including manufacturing orientation, workcenter, equipment, and weld sequence that specifies the details of how and where to manufacture.

The main object types shown in the assembly hierarchy are as follows:

Planning Object	Icon	Description
Block		High-level assembly that is a spatial definition of a section of the model structure. Blocks are created by splitting a volume and contain assemblies, parts, and/or child blocks created by splitting the block. Child of the top- level block (B0), an assembly block, or another block.
Assembly Block		Collection of blocks, assemblies, structural parts, equipment, outfitting parts, and outfitting assemblies (such as pipe spools). Child of the top-level block or another assembly block.
Assemblies		Collection of assemblies, structural parts, equipment, outfitting parts, and outfitting assemblies (such as pipe spools). Child of a block, assembly block, or another assembly.
Pipe spool		Assembly of pipe parts and components. Must consist of a minimum of one pipe part and one component (usually a flange). A spool usually runs between mechanical connections on a pipe run (for example, from flange to flange). Child of a block, assembly, assembly block, or the unassigned parts node.
UnAssigned Parts		Folder containing parts that have been automatically assigned to a block but have not been assigned to an assembly. Child of block node.
Parts	Same as **System** tab	Various light and detailed structural and outfitting parts created in other tasks. Child of a block, assembly, assembly block, or the unassigned parts node.
Planning Joint Folders		Folder containing Planning Joints. Child of a block, assembly, assembly block, or the unassigned parts node.
Planning Joints		Physical connection decomposed into its individual manufacturing components, such as the individual fillets on a double fillet weld. Child of the Planning Joint Folder.

NOTES

- Additional bitmap icons for user-defined assembly types are defined by and stored with the Planning reference data.
- Assembly and block icons can also be displayed with a warning icon  or an error icon  superimposed at the bottom right. These icons provide information that the **Check**

Manufacturability command has changed the status of the item due to less severe or severe detection of the manufacturability problems.

Icons in the Workspace Explorer

 Analysis Folder

 Analysis Model

 Assembly

 Assembly Connection

 Beam Part - undetailed

 Beam Part - detailed

 Beam Part - detailed, manufactured

 Bearing Plate - undetailed

 Bearing Plate - detailed

 Bearing Plate - detailed, manufactured

 Bearing Plate - group member

 Bearing Plate - group member, manufactured

 Bearing Plate - group master

 Bearing Plate - group master, manufactured

 Block

 Boundary Condition

 Bracket System or Leaf System

 Bracket Part - undetailed

 Bracket Part - detailed

 Bracket Part - detailed, manufactured

 Bracket Part - group member

 Bracket Part - group member, manufactured

 Bracket Part - group master

 Bracket Part - group master, manufactured

 Cable Nozzle on Equipment

 Cabletray Nozzle on Equipment

 Collar Plate - undetailed

 Collar Plate - detailed

 Collar Plate - detailed, manufactured

 Collar Plate - group member

 Collar Plate - group member, manufactured

 Collar Plate - group master

 Collar Plate - group master, manufactured

 Common Part Manager

 Conduit Nozzle on Equipment

 Coordinate System

 Coordinate System Axis

 Coordinate System Grid Plane

 Coordinate System Elevation Plane

 Coordinate System Radial Plane

 Coordinate System Radial Cylinder

 Curved Member Part - undetailed

 Curved Member Part - light

 Curved Member Part - light, manufactured

 Curved Member Part - detailed

 Curved Member Part - detailed, manufactured

 Curved Member Part - group member

 Curved Member Part - group member, manufactured

 Curved Member Part - group master

 Curved Member Part - group master, manufactured

 Designed Equipment Component

 Designed Equipment

 Designed Handrail

 Designed Member - undetailed

 Designed Member - light

 Designed Member - light, manufactured

 Designed Member - detailed

 Designed Member - detailed, manufactured

 Designed Member - group member

 Designed Member - group member, manufactured

 Designed Member - group master

 Designed Member - group master, manufactured

 Edge Treatment

 Equipment

 Equipment Component

 Equipment Foundation

 Equipment Shape

 Equipment Solid

 Equipment Solid - Added

 Equipment Solid - Subtracted

 Equipment Solid - Suppressed

 Feature (includes corner feature, edge feature, face feature, slot, and so on)

 Footing

 Footing Components

 Foundation Port

 Frame Connection

 Handrail

 HVAC Nozzle on equipment

 Insert Plate - undetailed

 Insert Plate - detailed

 Insert Plate - detailed, manufactured

 Insert Plate - group member

 Insert Plate - group member, manufactured

 Insert Plate - group master

 Insert Plate - group master, manufactured

 Ladder

 Linear Member Part - undetailed

 Linear Member Part - light

 Linear Member Part - light, manufactured

 Linear Member Part - detailed

 Linear Member Part - detailed, manufactured

 Linear Member Part - group member

 Linear Member Part - group member, manufactured

 Linear Member Part - group master

 Linear Member Part - group master, manufactured

 Load Case

 Load Combination

 Member Assembly Connection Cutback Feature

 Member Assembly Connection Plate Part

 Member Assembly Connection

 Member Fireproofing

 Member Part

 Member Split Connection

 Member System

 Opening

 Panel - manufactured

 Physical Connection

 Pipe Nozzle on Equipment

 Planning Joint - butt weld

 Planning Joint - lap weld

 Planning Joint - tee weld

 Planning Joint Folder

 Plate - light

 Plate Part - undetailed

 Plate Part - light

 Plate Part - light, manufactured

 Plate Part - detailed

 Plate Part - detailed, manufactured

 Plate Part - detailed, manufactured, with manufacturing part assembly child

 Plate Part - group member

 Plate Part - group member, manufactured

 Plate Part - group master

 Plate Part - group master, manufactured

 Plate System or leaf system

 Profile Edge Reinforcement System or Leaf System

 Profile Edge Reinforcement Part

 Profile Edge Reinforcement Part - light

 Profile Edge Reinforcement Part - light, manufactured

 Profile Edge Reinforcement Part - detailed

 Profile Edge Reinforcement Part detailed, manufactured

 Profile Edge Reinforcement Part - group member

 Profile Edge Reinforcement Part - group member, manufactured

 Profile Edge Reinforcement Part - group master

 Profile Edge Reinforcement Part - group master, manufactured

 Seam - Design

 Seam - Intersection

 Seam - Planning


Seam - Straking


Slab Assembly Connection


Slab


Stair


Standalone Beam Part - undetailed


Standalone Beam Part - detailed


Standalone Beam Part - detailed, manufactured


Standalone Plate Part - undetailed


Standalone Plate Part - detailed


Standalone Plate Part - detailed, manufactured


Standalone Plate Part - group member


Standalone Plate Part - group member, manufactured


Standalone Plate Part - group master


Standalone Plate Part - group master, manufactured


Standalone Profile Edge Reinforcement Part - undetailed


Standalone Profile Edge Reinforcement Part - detailed


Standalone Profile Edge Reinforcement Part - detailed, manufactured


Standalone Profile Edge Reinforcement Part - group member


Standalone Profile Edge Reinforcement Part - group member, manufactured


Standalone Profile Edge Reinforcement Part - group master


Standalone Profile Edge Reinforcement Part - group master, manufactured


Standalone Stiffener Part - undetailed


Standalone Stiffener Part - detailed

 Standalone Stiffener Part - detailed, manufactured

 Standalone Stiffener Part - group member

 Standalone Stiffener Part - group member, manufactured

 Standalone Stiffener Part - group master

 Standalone Stiffener Part - group master, manufactured

 Stiffener Part - undetailed

 Stiffener Part - light

 Stiffener Part - light, manufactured

 Stiffener Part - detailed

 Stiffener Part - detailed, manufactured

 Stiffener Part - group member

 Stiffener Part - group member, manufactured

 Stiffener Part - group master

 Stiffener Part - group master, manufactured

 Stiffener System or Leaf System

 Wall Assembly Connection

 Wall Part

 Wall Run

 Wall System

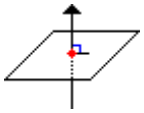
Plane Methods

You must often define planes used for object surfaces and sketching planes. Usually, but not necessarily, these planes are based from an existing plate or from a grid or reference plane created using the Grids task. The following methods define planes:

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

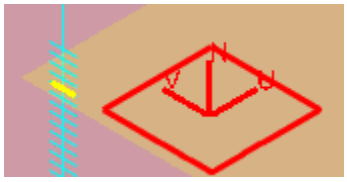
What do you want to do?

- *Define a coincident plane* (on page 25)
- *Define an offset plane* (on page 26)
- *Define a plane using a point and a normal vector* (on page 28)
- *Define a plane using three points* (on page 29)
- Move a defined plane

Define a coincident plane

1. Click **Offset from a Plane** .
2. In the model or **Workspace Explorer**, select a reference plane, a grid plane, or a planar plate system.

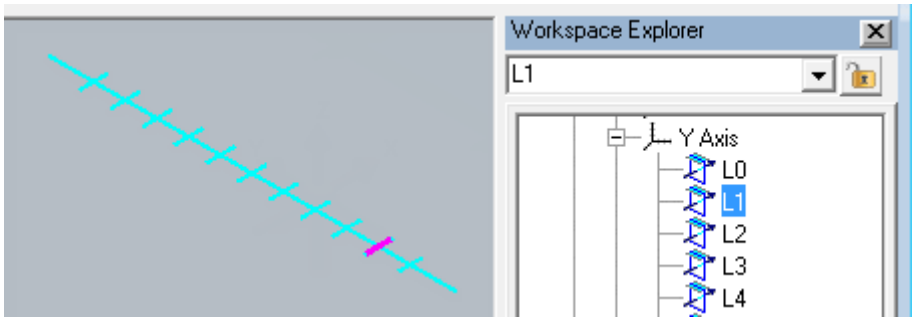
   A preview of the selected plane displays. The local coordinate system of the plane displays at the plane's center.



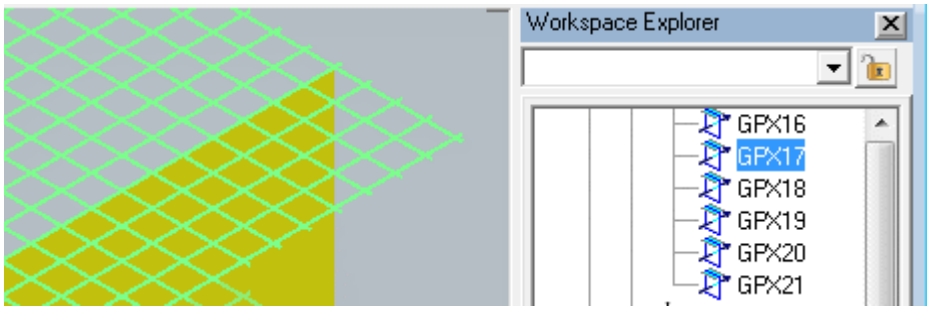
3. Check that **Offset Lock** is locked and that the **Offset** value is 0.

NOTES

- You create reference planes in the Grids task by selecting **Ship** in the **Grid Type** box of the **Grid Wizard**. For this plane method, you can select reference planes in the **Workspace Explorer** or in a graphic view.



- You create grid planes in the Grids task by selecting **Grids** in the **Grid Type** box of the **Grid Wizard**. For this plane method, you can best select grid planes in the **Workspace Explorer**.

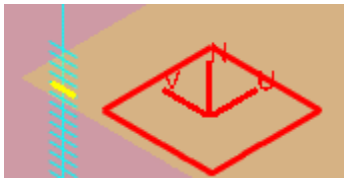


- For more information, see *Create Coordinate System (Grid Wizard)* in the *Grids User's Guide*.

Define an offset plane

1. Click **Offset from a Plane**.
2. In the model or **Workspace Explorer**, select the base plane from which to offset the new plane. The base plane can be a reference plane, a grid plane, or a planar plate system.

   A preview of the selected plane displays. The local coordinate system of the plane displays at the plane's center.



3. In the **Offset** box, type the offset distance from the plane and press TAB.

   The plane moves to the new location.

   -OR-

4. Graphically define the offset:
   a. Click **Offset Lock** .

      The option changes to unlocked .

   b. Move the pointer to the required location and click.

      The plane and plane coordinate system move to the new location, ***Offset Lock*** *changes to locked , and the offset value displays in the* ***Offset*** *box.*

   c. If an adjustment to the offset is needed, type a new value in the **Offset** box.
5. To move the plane independent of its previous definition, see Move a defined plane.

TIPS

- You can graphically define the offset with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.
- By moving the pointer back and forth over the base plane in the graphic view, you can change the offset direction. You can also change the direction by changing the sign in the **Offset** box.
- When **Offset Lock** is set to locked , the **Offset** value does not change when the pointer moves.

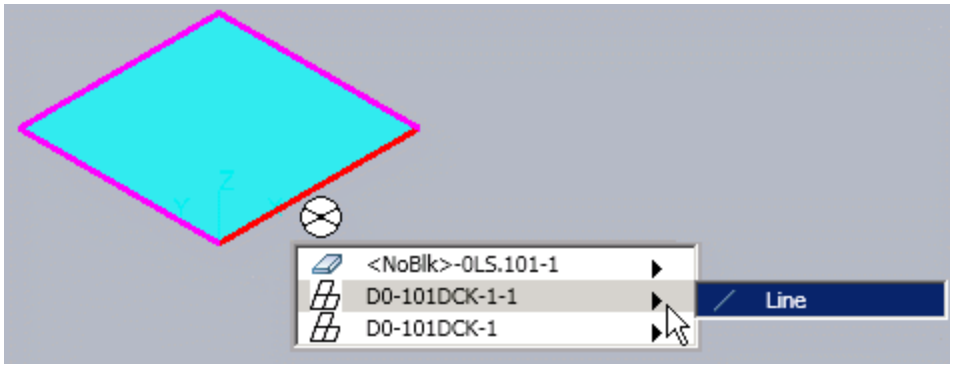
Define plane using angle from plane

1. Click **Angle from Plane** .
2. In the model or **Workspace Explorer**, select the base plane from which to angle the plane that you are defining. This plane can be a planar plate system, face port of a plate port, or a grid plane.
3. Select a rotation axis. The rotation axis must be parallel to the surface of the selected plane.

   TIP The rotation axis can be a linear seam, profile system, landing curve, system edge or connection.
4. In the **Angle** box, specify the rotation angle of the plane relative to the selected base plane.

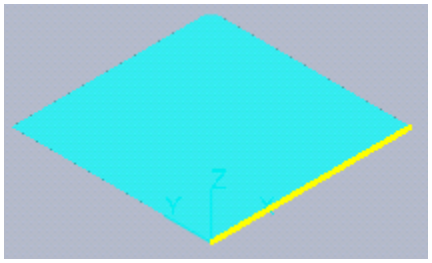
Define a plane using a point and a normal vector

1. Click **Plane by Point and Vector** .
2. Select a vector in the model that is normal to the required plane. The vector can be any linear geometry that is part of the construction of a model object, such as a plate system edge or a profile system landing curve. Use **QuickPick** to select a line geometry element.



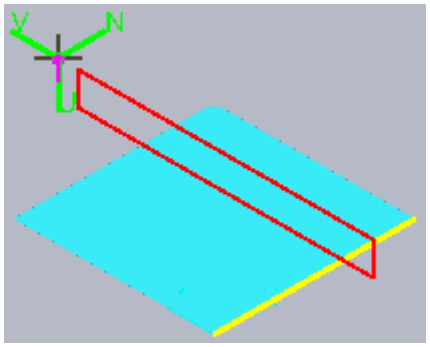
3. Click the **Line** geometry element.

   The selected geometry highlights in yellow.



4. Move the cursor to a point that is on the required plane, and then click.

   A preview of the plane and its local U_V_N coordinate system displays. The plane is coincident with the point and normal to the vector.



5. To move the plane independent of its previous definition, see Move a defined plane.

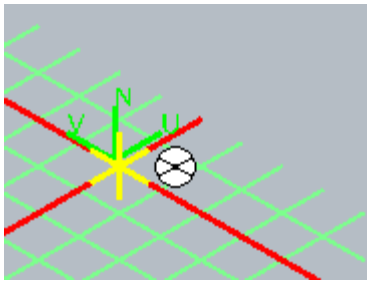
TIP You can graphically define the point with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.

Define a plane using three points

1. Click **Plane by Three Points** .
2. Specify on the first point that defines the plane.
3. Specify the second point that defines the plane.
4. Specify the third point that defines the plane.

NOTES

- You can define points on the surfaces, edges, and corners of systems and parts. You can control the types of points by using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.
- You can define points on grid planes, especially at intersections.



You can control the types of points using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

In Molded Forms, define parametric points by using **Insert** > **Topological Points**. For more information, see Topological Points (Insert Menu).

Define plane by vectors normal

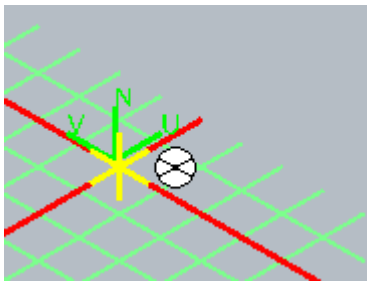
1. Click **Plane by Vectors Normal** .
2. In the model or **Workspace Explorer**, click the plane to which the new plane is normal.

   TIP The plane can be a planar plate system, a face port of a plate part, or a grid plane.
3. Specify the first point that defines the vector.
4. Specify the second point that defines the vector.

NOTES

- You can define points on the surfaces, edges, and corners of systems and parts. You can control the types of points by using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

- You can define points on grid planes, especially at intersections.



You can control the types of points using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

- In Molded Forms, define parametric points by using **Insert** > **Topological Points**. For more information, see Topological Points (Insert Menu).

Editing Properties

All items in the Planning task have properties that you can edit. Using the **Select** command on the vertical toolbar, select the item to edit.



An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains the available, predefined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered items in a graphic view and in the **Workspace Explorer**. For example, if you select **Blocks**, you can select only cutting blocks in the model or in the **Workspace Explorer**.

The Planning task includes these filters:

All Planning Items

Allows you to select blocks, cutting surfaces, or planning joints in the model and in the **Workspace Explorer**. Items placed using other tasks, such as pieces of equipment, cannot be selected using this filter.

Assemblies

Limits your selection in the model or in the **Workspace Explorer** to assemblies.

Assembly Blocks

Limits your selection in the model or in the **Workspace Explorer** to assembly blocks.

Blocks

Limits your selection in the model or in the **Workspace Explorer** to blocks.

Common Part

Parts that are processed by **Common Part Service** and associated with any common group are selected using this filter.

Common Part Group

Allows you to select common parts from **Common Part Groups** or assemblies under which the groups are created.

NOTE **Common Part Group** is not displayed in the **Workspace Explorer.** For more information, see *Select a common part group using filters* (on page 133).

Common Part Manager

Allows you to select **Common Part Managers** that are created under a block, assembly, or assembly block. You can select an assembly and navigate to the **Common Part Manager** associated to that assembly.

Construction Graphics

Limits the selection of items to construction graphics.

Control Points

Allows you to select all objects that are associated with it.

Cutting Surfaces

Limits your selection in the model or in the **Workspace Explorer** to cutting surfaces.

Geometric Constructions

Allows you to select all objects created using geometric constructions in the model or in the **Workspace Explorer**.

Part - Bracket

Allows you to select all the brackets in the model or in the **Workspace Explorer**.

Part - Collar

Allows you to select all the collars in the model or in the **Workspace Explorer**.

Part - Plate

Allows you to select all the plates in the model or in the **Workspace Explorer**.

Part - Profile

Allows you to select all the profiles in the model or in the **Workspace Explorer**.

Penetration Spools

Allows you to select all penetration spools that can include parts from one or more pipeline systems.

Planning Joints

Limits your selection in the model or in the **Workspace Explorer** to planning joints.

Planning Seams

Limits your selection in the model or in the **Workspace Explorer** to planning seams.

Spools

Allows you to select the objects that you want to spool.

All

Allows you to select any item, even items created in another task.

 Use the **Inside** fence command to select all items entirely inside the fence.

 Use the **Inside/Overlapping** fence command to select all items entirely inside the fence and those items outside but touching the fence at some point.

Common Property Tabs

The software displays some common property tabs on the properties dialog boxes for all planning items. Instead of repeatedly listing the common tabs with each equipment item property dialog box, they are documented here for easy reference.

See Also

Configuration Tab (on page 32)
Notes Tab (on page 33)
Relationship Tab (on page 34)
Occurrence Tab (on page 34)

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

General Tab

The **General** tab displays the properties that were selected by you or automatically determined by the software at creation. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

The properties displayed duplicate those available on other tabs, such as the **Main** tab. You can edit the properties on these tabs.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Occurrence Tab

Displays instance-specific information about the item.

Name - Displays the occurrence name of the item.

Property/Value table - Displays all the occurrence properties for the item as defined in the reference data. If the table is blank, the item for which you are viewing properties does not have any occurrence properties defined for it in the reference data.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.
- For plate and profile system part properties, parent systems are shown.
- For assembly connection properties, all connected objects are shown.
- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.
- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Permission Group Behaviors between Tasks

Permission groups provide controls for when, where, and by whom the model can be updated. Permission groups are assigned according to several common types of workflow. They typically fall into the following categories:

- By Function: such as Outfitting versus Structure, or Piping versus Equipment versus Molded Forms versus Structural Detailing.
- By System: such as Hull versus Deck1 versus TransvBhd1. This includes the ability to place the hull system (and its child leaf systems, seams, connections related seams, reference curves and openings) in a different permission group than the hull detailing objects and other connected objects.
- By Area: Fwd versus Aft, or by block or assembly.
- Combinations: for example, By Function and By Area (Molded Forms for Block1 versus Detailing for Block1).

The permission group of an object is not modified regardless of the **Active Permission Group** unless the permission group is explicitly changed on the **Configuration** tab of the **Properties** dialog box as described under the change propagation sections below.

Molded Forms Behavior

Plate systems and profile systems are assigned to the **Active Permission Group**.

Leaf systems, light parts, reference curves, and logical connections are assigned the permission group of their parent.

Seams and openings are assigned to the permission group of the plate or profile system they cut. You must have write access to the root plate system being cut when placing an opening or design seam.

In the **Execute Split** command, seams and knuckles that you do not have permission to modify are not selectable in the table.

Logical connections between profile systems and plate systems are in same permission group as the profile systems.

Logical connections between different root systems are created as children of the bounded or penetrated system.

Summary of Molded Forms permission behaviors:

Object	Permission Group	Comments
Root plate system	Active Permission Group	
Leaf plate system	Same as root plate system	
Root and child logical connections	Same as dependent root plate system	
Reference curve	Same as root plate system	
Design seam, intersection seam	Same as root plate system	Must have write access to root plate system
Opening	Same as root plate system	Must have write access to root plate system
Manual logical connection	Permission group of system for which you have write access, use first system selected if you have write access to both systems	Must have write access to at least one of the systems being connected
Root profile system	Active Permission Group	Profiles are handled similarly to plates
Logical connection between stiffener or edge reinforcement and the plate being stiffened	Same permission group as root profile system	
Beam	Active Permission Group	

Molded Forms Change Propagation

The root system and its child leaf system, logical connections, reference curves, seams and openings always have the same permission group. When the permission group is changed for any of these objects, the permission group is also changed for the other objects.

If plate system A is bounded by plate system B, you can modify A without write access to B. Plate System B is placed on the **To Do List** as **Out of Date** if the change to A impacts B.

Structural Detailing Behavior

The detailed part permission group is not assigned from the parent system permission group. Detailed parts created with **Execute Detailing** are assigned to the **Active Permission Group**. You can modify this assignment. You must have write permission to the part to detail the part.

If a detailed part is deleted, the resulting light part maintains the permission group of the detailed part.

You can assign different permission groups to parts sharing the same leaf system as created by a planning seam.

You can only add seams and features to a part if you have write permissions to the part.

Summary of Structural Detailing permission behaviors:

Object	**Permission Group**	**Comments**
Light Root Plate Part (before detailing)	Same as plate system at creation	
Detailed Root Plate Part	Active Permission Group when detailed. Part keeps this permission group if the detail part is deleted.	Must have write access to the light part
Leaf part	Same as root part	
Straking seam	Same as root part	Must have write access to root part
Feature placed by command (edge, corner, sketched, free edge treatment)	Same as root part	Must have write access to root part
Assembly Connection	Same as dependent part	
Feature, part, physical connection created by assembly connection	Same as assembly connection	
Free end cut object and its features	Same as root part	
Standalone part	Active Permission Group	

Structural Detailing Change Propagation

The detailed part, its child leaf parts, features and straking seams always have the same permission group. Assembly connections always have the same permission group as the dependent part in the connection. Assembly connection children always have the same permission group as the assembly connection. Standalone parts plus associated features and connections always have the same permission group. When the permission group is changed for any of these objects, the permission group is also changed for the other objects.

If you do not have write access to a part, the part is placed on the **To Do List** as **Out of Date** if a change is made to the part. Assembly connections and features are also not created until the part is updated on the **To Do List** by someone with write access.

Hole Management Behavior

Hole traces and outfitting catalog hole fittings are assigned to the **Active Permission Group**.

Hole cuts and structural hole fittings are assigned the permission group of their parent structural part (plate or profile).

Summary of Hole Management permission behaviors:

Object	Permission Group	Comments
Hole trace	Active Permission Group	
Hole cut	Same as root plate part	Must have write access to parent plate part. Root plate part must be detailed in the Structural Detailing task.
Outfitting catalog hole fitting	Active Permission Group	
Structural hole fitting	Active Permission Group	Must have write access to parent plate part. Root plate part must be detailed in the Structural Detailing task.

Hole Management Change Propagation

The permission group of an outfitting catalog hole fitting can be changed without changing the permission group of the hole trace.

The permission group of a structural hole fitting can be changed without changing the permission group of the hole cut or parent plate part.

A hole cut and its parent plate part always have the same permission group. When the permission group is changed for one of these objects, the permission group is also changed for the other object.

Planning Behavior

Planning permission group behavior is related to Molded Forms and Structural Detailing behavior because planning seams split leaf systems.

When **Manage Intersections** is run and an intersecting part is set to **Split** or **Offset**, planning seams are created in the **Active Permission Group**. You must have write access to the root system.

Summary of Planning permission behaviors:

Object	Permission Group	Comments
Planning seam	Active Permission Group	Must have write access to the root system of the parts being split.

Object	Permission Group	Comments
Block or assembly	Active Permission Group	

Planning Change Propagation

Write access is required to both parts to remove a planning split using **Flatten Assembly** or **Manage Intersections**. The resulting new part is assigned to the same permission group as the original parts if they belonged to the same permission group. The new part is assigned to the **Active Permission Group** if the original parts were in different permission groups.

Global Workshare Behavior

Global Workshare allows managed replication of the model at several sites. One of the restrictions of a Workshare configuration is that you only have Read access to objects that are controlled by non-local permission groups.

A system is a logical grouping of sub-systems. When you add or remove a sub-system, you are modifying the definition of the parent system. Therefore, you must have write access to the parent system. You do not need write access to the grandparent system.

When participating in a Global Workshare Configuration, you must manage all permission groups at the host site. The sub-system requirement to have write access to its parent system is not possible if the sub-system's permission group is created at the satellite site and the parent system's permission group is created at the host site.

For example, your host site is Houston and your satellite site is London. You create a system named "Midbody", and its controlling permission group is in Houston. You assign John, who works in London, write access. During the workshare replication process, the Midbody system and permission group is replicated in London. In London, John can add systems, plates, profiles and whatever else he wants to the Midbody system because John was given write access to the system's permission group in Houston. John cannot delete or change any of the properties of the Midbody system in London because the host site, Houston, owns it. He can only add objects to the system. If John were to travel to Houston and log on there, he could delete or change any of the properties of the Midbody system because the Houston host site owns it.

Split Notification

When a seam is added, deleted, or modified, the plate or profile systems and parts split by the seam are affected. The software notifies different tasks to transfer attributes and objects to the new systems and parts. The Molded Forms, Structural Detailing, Planning, Hole Management, and Structural Manufacturing tasks are affected by split notification.

The software uses split notification when:

- A design or planning seam is added and **Execute Split** is run in the Molded Forms task.
- A design or planning seam that has already been split is deleted or modified in the Molded Forms task.
- An intersection seam is added and split when **Execute Split** is run in the Molded Forms task.

- A profile or plate system creating an intersection seam is deleted or modified in the Molded Forms task.
- A planning seam is added when **Manage Block Intersections** is run and an intersecting part is set to **Split** or **Offset** in the Planning task.
- A planning seam is deleted when **Manage Block Intersections** is run and an intersecting part is set to **Intersected** or **Assigned** in the Planning task.
- A planning seam is modified when the cutting plane of a block is modified in the Planning task.

Identifying Seams in Workspace Explorer

You can identify the type of seam in the **Workspace Explorer** by the icon.

- Design seams
- Planning seams
- Straking seams
- Intersection seams

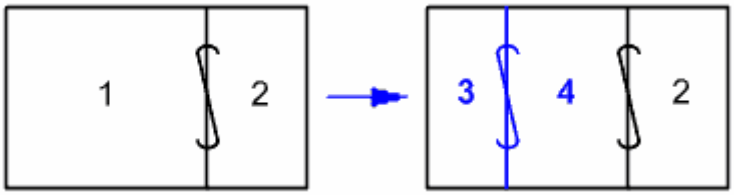
What do you want to do?

- *Add a seam* (on page 40)
- *Delete a seam* (on page 41)
- *Modify a seam when leaf systems are not added or removed* (on page 42)

Add a seam

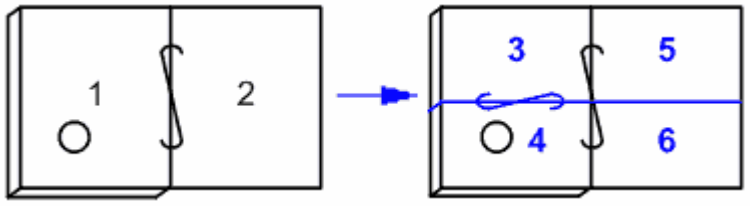
If you split a leaf system and its part with a new seam, the software creates new leaf systems and parts for each split, and deletes the original leaf system.

Molded Forms properties and material information are transferred to the new systems. The default naming rule is used to name the new leaf systems. Logical connections that cross the split are replaced by two new connections with the same properties.

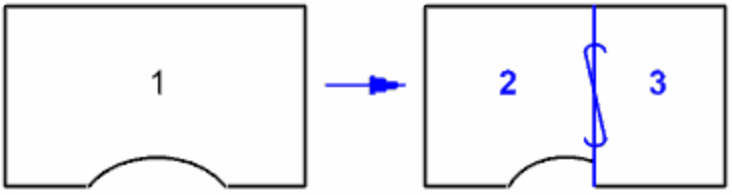


If the part is detailed, Structural Detailing properties and features are transferred to the new parts. The software creates new assembly and physical connections using properties from the previous connections.

⚠ CAUTION Straking seams, the leaf parts created by straking seams, and free edge treatments are not transferred and are deleted.

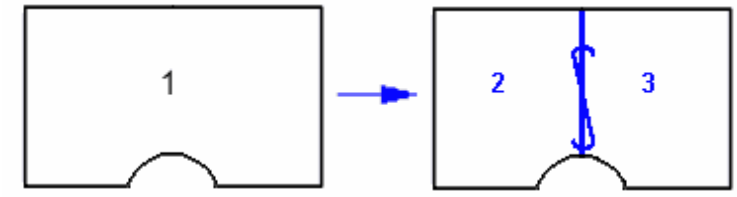


If a seam intersects an existing Structural Detailing feature, the feature is transferred to one of the parts.



Hole cuts and structural hole fittings in the Hole Management task are placed on the **To Do List**. Hole traces and outfitting catalog hole fittings are transferred to the new parts.

If a split passes through an edge feature definition point, the software creates two edge features. If the split does not pass through the definition point, then only the part containing the definition point contains an edge feature.



Block and assembly assignments in the Planning task are transferred. New parts are in the same assembly or block as the original part.

Manufacturing parts are updated using the **Manufacturing Service Manager**.

Connections to ladders, stairs, handrails, members, hangers, and equipment are transferred to the new part.

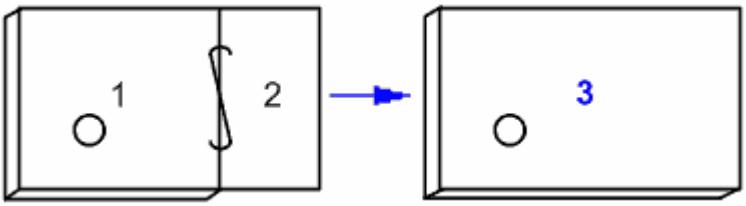
Delete a seam

If you delete or rebound a seam, the software deletes leaf systems and parts and creates new leaf system and parts.

Molded Forms properties and material information are transferred from the first system (as defined by the software) to the new system. The default naming rule is used to name the new leaf system. Logical connections that crossed the split are replaced by a new connection with the same properties as the first system.

Structural Detailing properties and features are transferred to the new part. If one part is detailed and the other is not, the new part is detailed.

⚠ CAUTION Straking seams, the leaf parts created by straking seams, and free edge treatments are not transferred and are deleted.



Hole cuts and structural hole fittings in the Hole Management task are placed on the **To Do List.** Hole traces and outfitting catalog hole fittings are transferred to the new parts.

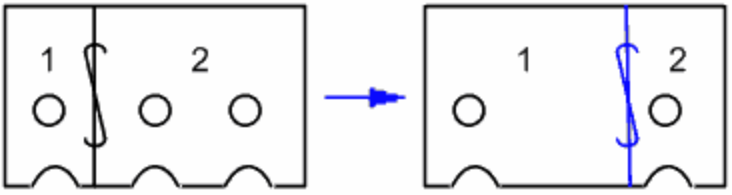
Block and assembly assignments in the Planning task are transferred. If the original parts were in the same assembly or block, then the new part is placed in that assembly or block. If the original parts were in different assemblies, then the new part is placed in the assembly last used by split notification.

Manufacturing parts are updated using the **Manufacturing Service Manager**.

Connections to ladders, stairs, handrails, members, hangers, and equipment are transferred to the new part.

Modify a seam when leaf systems are not added or removed

If you move a seam and the existing systems and parts are modified instead of replaced, then the software does not use split notification. **Structural Detailing** features are not transferred from one part to another, and are placed on the **To Do List**. If a feature is on one part and a seam is later modified so that it crosses the feature, the feature is still on one part. Split notification is not used when a seam is modified.



NOTES

- Split notification works with design seams from Molded Forms as well as with planning seams.
- If there are multiple features on one sketch, the software creates two sketches after the split; one for each part. Constraints associated with a sketched feature only go with one sketch after the split. You must manually add the constraints to the other sketch.

SECTION 2

Split Block

Designs new blocks by splitting existing blocks using the planar cutting planes. Initially, a single large block encloses the model. You use the **Split Block** command to gradually decompose this large block into smaller ones.

NOTES

- The splitting plane is infinite, but only splits the selected blocks.
- You can make the new blocks children of the original block by using the **Nest** option.
- You can make the new blocks consume the original block (that is, not children of the original) by using the **Flat** option.
- The software automatically assigns components that are entirely within the boundaries of a block to that block when the block assignment process is running.

Creating Blocks

A block is a high-level assembly that is a spatial definition of a section of the model structure created by splitting a volume. A block contains assemblies, parts, or child blocks created by splitting the block. A block can be a child of the top-level block (B0), an assembly block, or another block. A block can contain structural parts, outfitting parts, assemblies, and other blocks. The attached volume is closed and defined by a number of planar surfaces, referred to as *cutting planes*. Blocks always share their cutting planes with a neighboring block. If one of the cutting planes is moved, the block volumes that depend on that plane are modified. The definition and positioning of a cutting plane is based on existing structural parts or the reference system. Optionally, the cutting plane can be made associative with its base reference so that if a modification is made to the position of the base reference, the plane reflects those changes and the block is updated.

From a production viewpoint, model structure can be considered a collection, or assembly, of blocks and assemblies. However, in the design, model structure is a single block resulting from a top-down design process. This top-level block (B0) is automatically created by the Planning task. By default, the B0 encloses all model structure as defined by the bounding box of the structure.

Splitting the initial system-defined block with cutting planes creates new blocks. Further splitting and merging of blocks lets you create new blocks with a composite geometry.

When the block assignment process is running, the software automatically assigns any parts within the range of the block to that block. If you have the proper permissions, you can set up the process to run on a server, or the process can be run periodically on a client. The block assignment process listens for new and modified parts in the database, and puts them into the folder containing parts that have not yet been processed by the user of each block. For more information, see *Block Assignment* (on page 156).

You can also manually reassign the intersected parts to neighboring blocks if you need to by using the **Manage Block Intersections** command. You can reassign parts that are fully within the boundaries of a block by dragging the parts to any block or assembly in the **Workspace Explorer**. For more information, see *Block Assignment* (on page 156).

Top-down versus Bottom-up Planning

The default process for creating blocks is based on the concept of top-down planning. By using the **Nested** option in **Split Block**, you create a hierarchy of blocks starting with the top-most block (B0), the entire marine structure, and successively breaking the blocks into smaller volumes that are children to the larger blocks.

Blocks can also be created using a bottom-up planning concept. In this case, all of the blocks are created using the **Flat** option in **Split Block** so that all of the blocks are at the same level. The block hierarchy is then created by creating assembly blocks using **New (Block) Assembly**. In other words, blocks are initially broken into their smallest components at the bottom of the hierarchy, and combined upwards to create the large blocks.

At some point for both top-down and bottom-up planning, block definition stops and assembly definition begins using **New (Block) Assembly** . The point where this transition occurs is a function of the workflow of the facility and is not determined by the functionality of the software.

Split Block Ribbon

Sets options for splitting blocks in the model. The ribbon appears automatically after you click **Split Block**.

Select blocks to split

Specifies the block(s) to be split. Blocks can be selected from the model or from the **Workspace Explorer**. Multiple blocks can be selected sequentially.

Select cutting planes

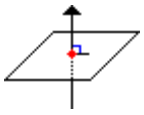
Defines the cutting planes using one of five methods: **Coincident**, **Offset from a Plane**, **Angle from Plane**, **Plane by Point & Vector**, **Plane by Three Points**, or **Plane by Vectors Normal**. Multiple planes are defined sequentially. You define the method of cutting plane definition for each plane you select.

Define Cutting Plane Controls

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a

graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**



Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Finish

Commits the new blocks to the database. The new blocks are created, the Split Block command is terminated, and the newly created blocks remain selected. You can further split the set of selected blocks by selecting the Split Block command again.

Reject

Clears the current block selection and starts over.

Accept

Accepts the current block selection.

Nest/Flat

Defines whether the new blocks will be nested. The selection you make applies to all the blocks defined during the command session. The **Flat** option causes the newly created blocks to replace the block from which they were created. The **Nest** option causes the newly created blocks to become children of the block from which they were created. The **Nest/Flat** option remains available until the **Finish** button is clicked.

Keep relationship

Creates an associative relationship between each plane and its base reference. When the option is checked, any subsequent modifications to the location of the base reference will also relocate the associated plane.

Edit Block Ribbon

Displays options for editing a block. The ribbon displays automatically when a block is selected in the graphic view or the **Workspace Explorer**.

Block Properties

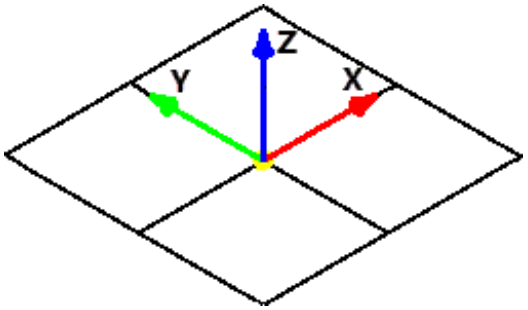
Activates the **Block Properties** dialog box. You can use this dialog box to modify properties such as assembly type or build method, which you cannot set on the ribbon. For more information, see *Block Properties Dialog Box* (on page 51).

Block Size

Allows the top-level B0 block to be set to a larger size than the default hull bounding box. This option is only available when the top-level B0 block is selected.

Manufacturing Orientation

Sets the manufacturing orientation and a viewing direction of the selected block. When a block is created, it is given a default manufacturing orientation of ship's position as defined by the global coordinate system. The manufacturing orientation symbol appears:



The x--y' plane created for manufacturing orientation controls the orientation of the block as it is manufactured on a shop floor.

TIPS

- Manufacturing orientation is also known as assembly orientation.
- X, Y, and Z are the global ship coordinates; x', y', and z' are the local manufacturing orientation coordinates.
- z' axis is the local manufacturing orientation up-vector. By default it is in the same direction as the global Z axis.
- Manufacturing orientation is used by *Show Assembly View* (on page 149), the **Structural Manufacturing** task, and the assembly method drawings in the **Ship Drawings** task.

When the manufacturing orientation is modified, the rotation of the local axes is recomputed according to the following factors:

a. The software determines which global axis, X, Y or Z, is closest to the local up-vector, z'.

b. If it is X, then the y'-axis is aligned to be as close as possible to Y.

c. If it is Y, then the x'-axis is aligned to be as close as possible to X.

d. If it is Z, then the x'-axis is aligned to be as close as possible to X.

Finish

Commits the changes to the database. The **Finish** button remains disabled until the block is modified in some way.

Cancel

Cancels the selections.

Accept

Accepts the selections.

Name

Displays the name of the selected block. This field is editable.

Manufacturing Orientation Controls

Orient by baseplate

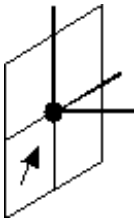
Allows you to define the manufacturing orientation of the selected item by selecting the face of a plate. The face of the plate becomes the x'-y' plane. If the selected plate is planar, the selected face is oriented horizontally (as on the workshop floor). If the selected plate is curved, the software sets that plate as the baseplate and then defaults to the 3-point method of definition for defining the up-vector.

Orient by 3 point plane

Allows you to define an x'-y' plane by three points, which in turn is used to define the z' manufacturing up-vector of the selected item. The defined plane is oriented horizontally (as on the workshop floor).

Edit view direction

Allows you to modify the view direction of the selected item. The view direction symbol is an arrow placed in the manufacturing orientation symbol. You can change the viewing direction to be used when viewing the assembly in manufacturing orientation and on building method drawings:



The x'-y' plane is divided into quadrants. An arrow shows the direction from which the object is viewed when viewing in manufacturing orientation and on assembly method drawings (isometric view from the indicated direction).

A default viewing direction is initially provided based on rules. When moving the mouse over one of the four base-plane quadrants, it highlights. Selecting one will swap the viewing direction and show the arrow in the selected quadrant.

Edit manufacturing orientation

Toggles the direction of the z ' manufacturing up-vector about the x'-y' plane used for definition of the orientation.

Edit Cutting Surface Ribbon

Displays options for editing a cutting surface. The ribbon displays automatically when a cutting surface is selected in the graphic view.

Select reference plane

Sets the reference plane for the cutting surface.

Select axis of rotation

Selects an axis of rotation if you want to angle the reference plane. The axis can be any linear edge, seam, or profile system that is parallel to the surface of the reference plane.

Keep Relationship

Creates an associative relationship between the cutting surface and its reference plane. When the option is checked, any subsequent modifications to the location of the reference plane will also relocate the cutting surface.

Offset

Specify the offset distance for the cutting surface about the reference plane.

Offset direction

Toggles the direction of the cutting surface about the reference plane. This option is only available if an **Offset** value is entered.

Angle

Specify the angle for the cutting surface. This option is only available when you use the

Select axis of rotation option.

What do you want to do?

- *Split a block* (on page 49)
- *Modify a block* (on page 49)
- *Modify the size of the top-level block (B0)* (on page 50)
- *Set block manufacturing orientation by baseplate* (on page 50)
- *Set block manufacturing orientation by three-point plane* (on page 51)

Split a block

1. Click **Split Block**  .
2. In the **Workspace Explorer**, select the block to split.
3. Click **Select Cutting Planes**  , and specify the cutting plane with which to split the block using one of the following methods:

   Define plane using angle from plane (on page 27)

   Define a coincident plane (on page 25)

   Define an offset plane (on page 26)

   Define a plane using three points (on page 29)

   Define a plane using a point and a normal vector (on page 28)

   Define plane by vectors normal (on page 29)
4. Click **Accept**  .
5. Set any additional block properties.
6. Repeat steps 3 and 4 to continue splitting the selected block into smaller sections.
7. Click **Finish**.

Modify a block

1. Click **Select**  on the vertical toolbar.
2. Select **Blocks** in the **Locate Filter** box.
3. Select the block to modify.
4. Using the controls on the **Edit Block** ribbon, modify the block as needed.

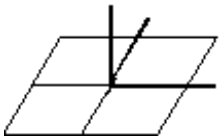
Modify the size of the top-level block (B0)

1. Click **Select** on the vertical toolbar.
2. Select **Blocks** in the **Locate Filter** box.
3. Select the top-level block **B0**.
4. Click **Block size**.
5. Select at least one plate system, plate sub-system, or grid plane outside of the existing **B0** bounding box.
6. Click **Finish**.

Set block manufacturing orientation by baseplate

1. Select an existing block from a graphic view or from the **Workspace Explorer**.
2. Click **Orient by baseplate** on the ribbon.
3. Select the plate in the assembly that you want to function as the manufacturing orientation baseplate

   A graphic of the x'-y' plane and the z' up-vector direction appears on the selected plate in the graphic view.



4. If required, click **Edit manufacturing orientation** to toggle the direction of the z' up-vector.
5. If required, click **Block Properties** to further edit the manufacturing orientation, including rotation about each coordinate system axis. For more information, see *General Tab (Block Properties Dialog Box)* (on page 52).
6. Click **Finish**.

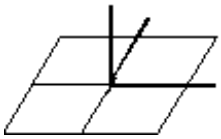
   NOTE *You cannot set the manufacturing orientation when creating new blocks with the* ***Split Block*** *command. You must select an existing block and modify it.*

Set block manufacturing orientation by three-point plane

1. Select an existing block from a graphic view or from the **Workspace Explorer**.
2. Click **Orient by 3 point plane** on the ribbon.
3. Select three points in the mode that form the plane to use for the manufacturing orientation.

   TIP You can use **PinPoint** to define the three points.

   A graphic of the x'-y' plane and the z' up-vector direction appears on the selected plate in the graphic view.

4. If required, click **Edit manufacturing orientation** to toggle the direction of the z' up-vector.
5. If required, click **Block Properties** to further edit the manufacturing orientation, including rotation about each coordinate system axis. For more information, see *General Tab (Block Properties Dialog Box)* (on page 52).
6. Click **Finish**.

   NOTE You cannot set the manufacturing orientation when creating new blocks with the **Split Block** command. You must select an existing block and modify it.

Block Properties Dialog Box

Specifies the properties for the block you are editing and includes the following tabs:

General Tab (Block Properties Dialog Box) (on page 52)
Relationship Tab (on page 34)
Configuration Tab (on page 32)
Notes Tab (on page 33)

See Also

Modify the size of the top-level block (B0) (on page 50)
Modify a block (on page 49)

General Tab (Block Properties Dialog Box)

Displays and defines the general property names and values of the selected block. Only information that appears in a field with a white background may be edited. All other fields are read-only.

Category

Select to display a group of block properties. Available settings are **Standard**, **Planning**, **Weight and CG**, and **Production**.

Update

Calculates the non-persisted properties, such as length and weight.

Standard Category

Name

Name of the block. This name is usually generated by the active name rule. You can type a different name for the block if you want. Names generated by a rule include a Global Workshare name rule ID, if the name rule ID was defined when the model database was created. For more information, see "Using Global Workshare" in the *Global Workshare Guide*.

Name Rule

Name rule used to generate the name. This field allows you to have optional rules for different situations. The content of the list comes from the "Rule Options" function. Set this option to **User Defined** if you want to specify the name yourself. Names generated by a rule include a Global Workshare name rule ID, if the name rule ID was defined when the model database was created. For more information, see "Using Global Workshare" in the *Global Workshare Guide*.

Assembly Type

Specifies the type of planning assembly, such as **block**, **assembly**, or **pipe spool**. The list contains all available assembly types, as defined in the reference data. The list is defined by the AssemblyType codelist.

Parent Assembly

Displays the name of the parent assembly, block, or assembly block.

Length Actual

Displays the length dimension along the X-axis of all parts and assemblies (range box dimensions) in the block relative to manufacturing orientation and the **Manufacturing Frame System**, regardless of the **Length Nominal** value of the block. Excludes any parts in the **UnAssigned Parts** folder.

Width Actual

Displays the width dimension along the Y-axis of all parts and assemblies (range box dimensions) in the block relative to manufacturing orientation and the **Manufacturing Frame System**, regardless of the **Length Nominal** value of the block. Excludes any parts in the **UnAssigned Parts** folder.

Height Actual

Displays the height dimension along the Z-axis of all parts and assemblies (range box

dimensions) in the block relative to manufacturing orientation and the Manufacturing Frame System, regardless of the Length Nominal value of the block. Excludes any parts in the **UnAssigned Parts** folder.

Length Nominal

Displays the length dimension along the X-axis of the manufacturing orientation range box. The manufacturing orientation range box is determined by the dimensions of the block relative to the manufacturing orientation and the **Manufacturing Frame System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

Width Nominal

Displays the width dimension along the Y-axis of the manufacturing orientation range box. The manufacturing orientation range box is determined by the dimensions of the block relative to the manufacturing orientation and the **Manufacturing Frame System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

Height Nominal

Displays the height dimension along the Z-axis of the manufacturing orientation range box. The manufacturing orientation range box is determined by the dimensions of the block relative to the manufacturing orientation and the **Manufacturing Frame System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

Assembly Stage

Specifies the construction stage of a block from a facility-defined rule set. The list contains all available stages as defined in the reference data. Examples of stages include **part production**, **assembly of subassemblies**, **assembly of sections with outfitting**, and **assembly of sections before painting**. If no stage is defined, **<None>** is displayed. The list is defined by the AssemblyStage codelist.

Workcenter

Assigned assembly workcenter from a facility-defined rule set. The drop-down list contains all available workcenters regardless of their level in the workcenter hierarchy. Examples of workcenters include docks, shops, bays, and panel lines. Workcenters may also be areas and zones where a block is assembled from multiple assemblies.

Default Build Equipment

Primary equipment used to join the elements of the block. The drop-down list contains all available build equipment as defined in the reference data. Examples of build equipment include jig floors and bending equipment. Build equipment can be changed at the block element level by clicking **Sequence Form**  on the **Sequence** ribbon. For more information, see *Sequence* (on page 79).

Default Weld Equipment

Weld equipment used within the assigned workcenter when weld is defined using *Generate Planning Joints* (on page 90)  . The drop-down list contains all available weld equipment as defined in the reference data.

Exit Equipment

Crane or transport equipment within the assigned workcenter that moves the finished block

to its next assembly stage and workcenter. The drop-down list contains all available exit equipment as defined in the reference data.

Building method comments

Specifies comments regarding the workcenter and work process assignment or any other issue pertinent to the block.

Build And Weld As Sequenced

Specifies if you want to weld objects as they are joined in the assembly sequence. Select **True** to weld each object as it is joined, and select **False** to join all objects together before welding.

Orientation Baseplate

Displays the assigned manufacturing orientation baseplate if one has been defined using the **Manufacturing orientation** option. If no baseplate is defined, **<None>** is displayed.

Orientation Coordinate System

Allows you to select the global coordinate system used to define the local manufacturing orientation coordinate system in the **Manufacturing orientation** option. The drop-down list contains all available coordinate systems as defined in the reference data. Default value is the main coordinate system.

Orientation X Rotation

The rotation of the block around the X-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation** option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the block is no longer oriented by the baseplate.

Orientation Y Rotation

The rotation of the block around the Y-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation** option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the block is no longer oriented by the baseplate.

Orientation Z Rotation

The rotation of the block around the Z-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation** option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the block is no longer oriented by the baseplate.

Boardside

Board of the block. The drop-down list contains all available values as defined in the reference data, typically **Center**, **Starboard Side**, **Port Side**, and **Both Sides**. Default value is **Center**.

Planning Category

Build Method

The method used to position child objects in the block. The drop- down list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**.

Slot Connectivity

The minimum welding required at profile and slot intersections to meet build method requirements. The drop-down list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**.

TIP You can customize slot and collar rules in the Structural Detailing task to use **Build Method** and **Slot Connectivity** properties to determine the slots and collars placed during detailing.

Weight and CG Category

Weight Block

Displays the weight of all parts that are children to the block, including parts assigned to all assemblies that are children to the block, and the content of the **UnAssigned Parts** folder.

Weight unassigned parts

Weight of all parts in the **UnAssigned Parts** folder of the block.

Weight Intersecting

Weight of all parts with **Intersected** status with cutting planes of the block, as defined in the *Manage Block Intersections Dialog Box* (on page 62).

CoG X

Displays the position of center of gravity along the X-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the Model.

CoG Y

Displays the position of center of gravity along the Y-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the Model.

CoG Z

Displays the position of center of gravity along the Z-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the Model.

CoG Coordinate System

Allows you to select the coordinate system for calculation of center of gravity (CoG) coordinates. The drop-down list contains all available coordinate systems as defined in the reference data. Default value is the main coordinate system.

Production

Manufacturing Frame System

Allows you to select the coordinate system used to process structural manufacturing rules for all objects in the block. The drop-down list contains all available coordinate systems as defined in the reference data. If no coordinate system is defined, **No Framesystem** is displayed and the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation** option) is used.

X-Frames

Specifies the setting for the frames along the X-axis in the manufacturing frame system. Click the **Value** field to the right of **X-Frames** to display the **Select or Modify Frames** *dialog box* (on page 56).

Y-Frames

Specifies the setting for the frames along the Y-axis in the manufacturing frame system. Click the **Value** field to the right of **Y-Frames** to display the **Select or Modify Frames** *dialog box* (on page 56).

Z-Frames

Specifies the setting for the frames along the Z-axis in the manufacturing frame system. Click the **Value** field to the right of **Z-Frames** to display the **Select or Modify Frames** *dialog box* (on page 56).

See Also

Modify the size of the top-level block (B0) (on page 50)
Modify a Block (on page 49)
Select or Modify Frames Dialog Box (on page 56)
General Tab (Block Properties Dialog Box) (on page 52)

Select or Modify Frames Dialog Box

Defines frames for the manufacturing frame system.

Coordinate System

Allows you to select a coordinate system from a list of coordinate systems available in the model.

Axis

Defines the axis of the frames.

Available Frames

Displays a list of the frames that are in the block range. You can select multiple frames from the list to add to **Selected Frames**.

Add

Adds frames to **Selected Frames**.

Remove

Removes frames from **Selected Frames**.

Selected Frames

Displays the frames that you have selected.

OK

Applies the settings. The selected frame names will be shown in the corresponding property fields in the **General** tab of the **Block Properties** dialog box.

See Also

General Tab (Block Properties Dialog Box) (on page 52)

SECTION 3

Manage Block Intersections

Identifies the parts that intersect with a block cutting plane, define whether to assign the part to a block or where to split the intersecting structural parts by perpendicular offset from the cutting plane, and then create the split. Parts are initially assigned as defined by the **Block Assignment** process.

Anyone with write access permission to the part items is allowed to create splits using **Manage Block Intersections** . If you do not have write permissions to the items, the items can be assigned to a block but cannot be split. In the absence of write access permissions, if you create or edit a split or move a block cutting plane, the associative edits of the splits appear on the **To Do List** (click **View > To Do List**).

Assigning Parts to Blocks and Assemblies

You have several methods for assigning parts to blocks or assemblies:

- Using *Manage Block Intersections* (on page 58) for most block assignments.
- Using *New (Block) Assembly* (on page 64) for initial assembly and assembly block assignments.
- Using a drag-and-drop operation in the **Workspace Explorer** for additional or modified block, assembly, or assembly block assignments.

Manage Block Intersections identifies the parts that intersect with a block cutting plane, defines whether to assign or split a part, defines where to split the intersecting part (by perpendicular offset from the cutting plane), and (if you have write permission for the item) creates the split. The software associates splits to block cutting planes. When you move or delete planes, you move or delete the splits as well. Parts are initially assigned as defined by *Block Assignment* (on page 156).

The software lists the intersections of parts with block cutting planes. If a part has multiple intersections with the selected cutting planes, the software lists multiple intersections. You can define the offset and direction to modify the position of each intersection individually. This modifies the position of the split. After you cut the parts, the software keeps track of splits that you created.

The intersections appearing on the **Manage Block Intersections** dialog box can be controlled by selecting one or more parts, cutting planes, or blocks.

When using **Manage Block Intersections** , you cannot reassign parts that are fully within the boundaries of a block. A part completely within the boundaries of a block may need to be assigned to another block due to construction sequencing, most commonly for parts near block boundaries. In this case, you can reassign parts that are fully within the range of a block by dragging the parts to any block or assembly in the **Workspace Explorer**.

You can also use the drag-and-drop operation to reassign parts in assemblies and assembly blocks.

Planning Seams

A planning seam is created when **Manage Block Intersections** is run and an intersecting part is set to **Split** or **Offset**.

A planning seam is deleted when **Manage Block Intersections** is run and an intersecting part is set to **Intersected** or **Assigned**.

A planning seam is modified when the cutting plane of a block is modified.

NOTE Planning seams can also be added manually using the seam commands in the Molded Forms task. For more information, see *Creating Seams* in the *Molded Forms User's Guide*.

Identifying Seams in Workspace Explorer

Identifying Seams in Workspace Explorer

You can identify the type of seam in the **Workspace Explorer** by the icon.

- Design seams
- Planning seams
- Straking seams
- Intersection seams

Intended Use of Seams

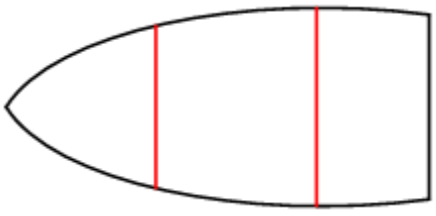
Intended Use of Seams

Intersection Seams

Created in the Molded Forms task at the intersection of two parts, such as where a deck and a bulkhead intersect. Intersection seams are automatically created by the Execute Split command in Molded Forms.

Design Seams

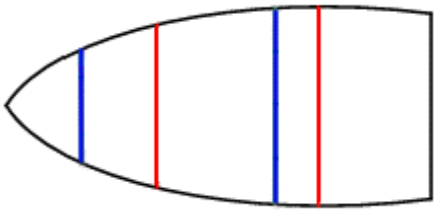
Created in the Molded Forms task using a seam command. A design seam is used where a design property of a system is changing, such as where the thickness of a plate changes. Leaf systems under a single parent system are created.



Planning Seams

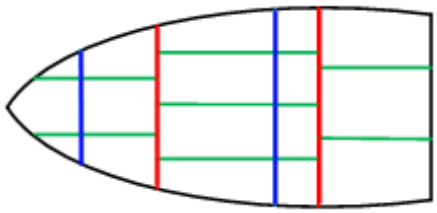
Create in the Planning task using **Manage Block Intersections** or in the Molded Forms task using a seam command. A planning seam designates the limits of blocks (large segments

of a ship assembled in a single location) or smaller assemblies when the limits do not coincide with design seams.



Straking Seams

Created in the Structural Detailing task to designate separate physical plates, usually due to restrictions in plate sizes that can be ordered from the mill. Plate parts under a single leaf system are created.



Manage Block Intersections Ribbon

Sets options for managing block intersections in the model. The ribbon displays automatically after you select **Manage Block Intersections** .

Select Intersections

Allows you to select elements to be queried for intersections. You can select any of the following: **Parts**, **Cutting Planes**, or **Blocks**.

Manage Block Intersections

Activates the **Manage Block Intersections** dialog box. For more information, see *Manage Block Intersections Dialog Box* (on page 62).

Cancel

Clears the current selections and starts over.

Accept

Accepts all the current selections.

Select by

Controls the method of query for intersections. At least one of the three **Select by** options must always be checked.

- **Parts** - Indicates that only parts are selectable and that all intersections of the selected parts must be listed on the **Manage Block Intersections** dialog box. You can use the **Fence** command to "fence in" the parts you want to select in the graphic view.
- **Cutting Planes** - Indicates that only cutting planes are selectable and that all intersections with the selected cutting planes must be listed in the **Manage Block**

Intersections dialog box. Selecting by **Cutting Planes** and **Blocks** means that only cutting planes and blocks are selectable and that all intersections with the selected cutting planes and blocks are listed in the dialog box.

- **Blocks** - Indicates that only blocks are selectable and that all intersections with the selected blocks must be listed on the **Manage Block Intersections** dialog box. Selecting by **Cutting Planes** and **Blocks** means that only cutting planes and blocks are selectable and that all intersections with the selected cutting planes and blocks are listed in the dialog box.

TIPS

- Selecting by **Cutting Planes** and **Blocks** means that only cutting planes and blocks are selectable and that all intersections with the selected cutting planes and blocks are listed in the dialog box.
- Selecting by **Parts** along with cutting planes or **Blocks** allows you to select parts and cutting planes or blocks; however, all intersections between the selected parts and the selected cutting planes or blocks must be listed in the dialog box. If no cutting planes or blocks have been selected, all intersections of the selected parts are listed.

Show intersections

Specifies whether or not structural or outfitting intersections are listed.

- **Structural** - Specifies that structural intersections be listed on the **Manage Block Intersections** dialog box.
- **Outfitting** - Specifies that outfitting intersections be listed on the **Manage Block Intersections** dialog box.

What do you want to do?

- *Manage intersections* (on page 61)
- *Use a drag-and-drop operation to assign parts* (on page 62)

Manage intersections

1. Click **Manage Block Intersections** on the vertical toolbar.
2. If necessary, change the **Select by** setting to select parts, cutting planes, or entire blocks.
3. Click **Cutting Planes** on the toolbar, and select the cutting plane or block for which you want to manage the intersections.
4. If necessary, change the **Show intersections** setting to restrict the type of parts for which you want to manage intersections.
5. Click **Manage Intersections** on the ribbon, and set the intersection parameters to meet your needs. Change the value in **Status** first, then the value in **Offset** and finally the value in **Into Block**. Click **Show** to fit the selected intersection to the active graphics view.
6. Click **OK**, **Apply** or **Cancel** to close the dialog box.

Use a drag-and-drop operation to assign parts

1. On the **Assembly** tab of the **Workspace Explorer,** select one or more parts.

   NOTE Parts at any level of the **Assembly** tab hierarchy can be selected.

2. Drag the parts to the needed block, assembly, or assembly block.

Manage Block Intersections Dialog Box

Lists the intersections for the selected parts, cutting planes, or blocks. Options in the dialog box allow you to change the intersection to **Split** (for structural parts only) or to **Assign** the intersected part to a block. You can also specify an offset and a direction to modify the position of the split. After parts are cut, the dialog box keeps track of splits that the **Manage Block Intersections** command created. If parts resulting from a planning split are later split from within another task (such as Structural Detailing), the dialog box will continue to keep track of the original higher level splits done.

Identification

Displays the name of the intersected part. Placing a split that divides the initial part into more than two parts adds relations on the intersection object to the new parts. The **Manage Block Intersections** dialog box keeps track of the initial intersection.

Status

Controls the allowed/available settings for **Offset** and **Into Block**. The following status types are available:

- **Intersected** - The identified part intersects the cutting plane. This is the default status. The **Offset** and **Into Block** fields remain empty.
- **Split** - Creates a split in the position indicated by offset and offset direction. Only intersecting structural parts can be split. This status is unavailable if a component, such as an outfitting part, cannot be split.
- **Assigned** - Assigns the entire identified part to the indicated block without splitting it.

Offset

Allows you to key in an offset value. An offset can only be set if the status is set to **Split**. When the status is changed to **Assigned** or back to **Intersected**, the **Offset** field is cleared. Additional parts are created as the result of a split and are automatically assigned to either block dependent on the offset location.

Into Block

Specifies the block into which the offset split is applied. This option is only available when there is an offset value for a split.

Multiple selection intersection edit

Specifies the **Status**, **Offset**, and **Into Block** settings to apply at one time to multiple selected items in the list.

Information on selected intersection

Provides details on **Blocks** and **Parts** involved in the intersection of a selected item in the

list.

Show

Highlights in the model the blocks and parts selected.

Planning Seams

A planning seam is created when **Manage Block Intersections** is run and an intersecting part is set to **Split** or **Offset**.

A planning seam is deleted when **Manage Block Intersections** is run and an intersecting part is set to **Intersected** or **Assigned**.

A planning seam is modified when the cutting plane of a block is modified.

NOTE Planning seams can also be added manually using the seam commands in the Molded Forms task. For more information, see *Creating Seams* in the *Molded Forms User's Guide*.

SECTION 4

New (Block) Assembly

Creates a new assembly consisting of the parts or other assemblies you select in the graphic view or the **Workspace Explorer**. You can also use the command to create a new assembly block consisting of the parts, assemblies, or blocks you select in the graphic view or the **Workspace Explorer**.

Creating Assemblies

You can think of an assembly as a folder containing other assemblies and parts. To create a new assembly, you must select an existing assembly or block to act as the parent. You can also select parts to assign to the assembly. After you create the new assembly, you can make any changes you require (such as properties, assembly content, or orientation) and save the new assembly to the hierarchy. After you create the assembly, you can change assembly content from the **Workspace Explorer** using drag-and-drop, or you can use the **Cut** and **Paste** commands. **Cut** and **Paste** are available both from the **Edit** menu and the shortcut menu of the **Workspace Explorer**.

★ IMPORTANT **Paste** is available for block, assembly, and assembly block objects; **Cut** is only available for assembly objects.

Top-down versus Bottom-up Planning

The default process for creating blocks is based on the concept of top-down planning. By using the **Nested** option in **Split Block**, you create a hierarchy of blocks starting with the top-most block (B0), the entire marine structure, and successively breaking the blocks into smaller volumes that are children to the larger blocks.

Blocks can also be created using a bottom-up planning concept. In this case, all of the blocks are created using the **Flat** option in **Split Block** so that all of the blocks are at the same level. The block hierarchy is then created by creating assembly blocks using **New (Block) Assembly**. In other words, blocks are initially broken into their smallest components at the bottom of the hierarchy, and combined upwards to create the large blocks.

At some point for both top-down and bottom-up planning, block definition stops and assembly definition begins using **New (Block) Assembly** . The point where this transition occurs is a function of the workflow of the facility and is not determined by the functionality of the software.

New (Block) Assembly Ribbon

Assembly Properties

Activates the **Assembly Properties** dialog box. You can use this dialog box to modify properties such as assembly type or default build equipment, which you cannot set on the ribbon. For more information, see *Assembly Properties Dialog Box* (on page 72).

Selects Sub-assemblies and Parts

Select the assembly or assembly block content. You can select objects in either a graphic

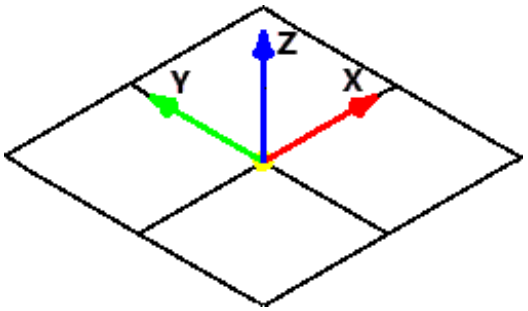
view or the **Workspace Explorer**. You can use the **Tools > Hide** command to hide the assembly content already selected, thus making it possible to find smaller objects.

Add Secondary Objects

Opens the **Secondary Objects** dialog box. In the dialog box, select the type of secondary objects to include. The secondary objects, such as collars and profiles, are on plates included in the assembly or assembly block.

Manufacturing Orientation

Sets the manufacturing orientation and a viewing direction of the selected assembly or assembly block. When an assembly or assembly block is created, it is given a default manufacturing orientation of marine structure's position as defined by the global coordinate system. The manufacturing orientation plane displays as follows:



The local x-y plane created for manufacturing orientation defines the orientation of the block as it is manufactured on a shop floor. The local z axis is the manufacturing orientation up-vector. By default, it is in the same direction as the global coordinate Z axis.

TIPS

- Manufacturing orientation is also known as assembly orientation.
- Manufacturing orientation is used by *Show Assembly View* (on page 149), the **Structural Manufacturing** task, and assembly method drawings in the **Ship Drawings** task.

See the *Manufacturing Orientation Controls* section below.

Finish

Commits the new assembly or assembly block to the database.

Cancel

Cancels the assembly content defined when the **Selects subassemblies and parts** step is active. When the **Select parent assembly or block** step is active, cancels the assembly parent. **Cancel** remains disabled until an assembly or part is selected.

Accept

When the **Selects subassemblies and parts** step is active on the ribbon, clicking **Accept** accepts the assembly content and moves on to the next step. When the **Select parent assembly or block** step is active on the ribbon, clicking **Accept** accepts the selected assembly parent and moves on to the next step. An accept command is needed in order to get a graphical view of the selected parent assembly. **Accept** remains disabled until an assembly or part is selected

Type

Specifies whether the new planning item is an assembly or an assembly block.

Name

Specifies the name of the new assembly or assembly block.

Parent Assembly

Allows you to select the parent assembly or block. Click **More** to choose another assembly or block with the **Select Assembly** dialog box.

Manufacturing Orientation Controls

Orientation Method

Provides the following options: **Global**, **User Defined**, or **By Object**.

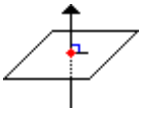
NOTE When the manufacturing orientation is modified, the rotation of the local axes is recomputed according to the following:

1. The software determines which global axis, the X, Y or Z, is closest to the local up-vector, z.
2. If it is X, then the y-axis is aligned to be as close as possible to Y.
3. If it is Y, then the x-axis is aligned to be as close as possible to X.
4. If it is Z, then the x-axis is aligned to be as close as possible to X.

Global - Orients the assembly according to the global coordinate system (default).

User Defined - Defines the orientation by selecting or creating a plane using one of the following six methods:

- **Coincident** - Defines a plane coincident to another plane.
- **Offset from Plane** - Defines a plane at a specified offset distance from another plane.
- **Angle from Plane** - Defines a plane at a specified angle or slope to another plane. You must define an axis of rotation and the angle or slope.
- **Plane by Point and Vector** - Defines a plane using a vector normal to the plane being defined and a third point to define the plane position along the vector.



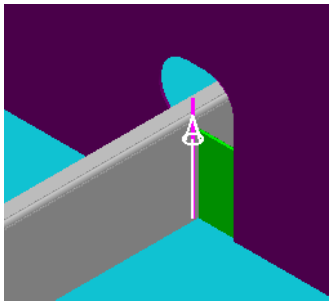
- **Plane by Three Points** - Defines a plane using three points that you identify in the model.
- **Plane by Vectors Normal** - Specifies the reference plane as being normal to another plane that you select and having a rotation parallel to a vector that you define.
- **Remove/Maintain Relationships of the Plane -** Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

By Object - Defines the orientation by specifying a type of object from the **Object Type** list, and then selecting the object in a graphic view or in the **Workspace Explorer**. The supported object types are listed below.

Object Type - Specifies the type of object used to define the orientation.

- **Plate** - Defines the manufacturing orientation of the assembly by selecting the face of a plate. The plate becomes the base plate, the face of the plate becomes the x'-y' plane and the orientation up-vector matches the z' axis. If the selected plate is planar, the selected face is oriented horizontally. If the selected plate is curved, the software sets that plate as the base plate and then defaults to the 3-point method of definition for defining the up-vector. The plate must be a child of the assembly.
- **Pin Jig** - Defines the manufacturing orientation of the assembly by selecting a pin jig. The base plane of the pin jig becomes the x'-y' plane and the orientation up-vector matches the z' axis. The pin jig must be related to one of the children of the assembly.
- **Intersection Seam** - Defines the manufacturing orientation of the assembly by selecting an intersection seam. The orientation up-vector (the z' axis) matches the direction of the seam. This method only allows selection of intersection seams between a plate and a profile. The plate and profile pair that define the seam must be children of the assembly.



- **Seam (By Rule)** - Defines the manufacturing orientation of the assembly by selecting an intersection seam. The orientation up-vector (the z' axis) matches the direction of the seam. This method only allows selection of intersection seams between a plate and a profile that lead to slots on the plate after detailing in the Structural Detailing task. The plate and profile must be children of the assembly and the slot must have a positive opening angle.
- **Coordinate System** - Defines the manufacturing orientation of the assembly by selecting a coordinate system. The orientation up-vector (the z' axis) matches the direction of the coordinate system Z-axis.
- **Assembly** - Defines the manufacturing orientation of the assembly by selecting a sub-assembly (that is, a child assembly in the **Assembly** tab hierarchy of the Workspace Explorer). The orientation up-vector (the z' axis) matches the direction of the up-vector of the sub-assembly.

- **Frame/Axis** - Defines the manufacturing orientation of the assembly by selecting an axis of a coordinate system. The orientation up-vector (the z' axis) matches the direction of the axis.
- **Plane** - Defines the manufacturing orientation of the assembly based on the normal direction of the selected plane. A plane is a grid plane or a planar surface on an object, such as a plate.

Orientation Object - When the **Orientation Method** is set to **By Object**, the list of available of objects that can be used to define the orientation is displayed. You can also select one of these objects from the graphic view or the Workspace Explorer.

NOTE **Orientation Objects** is not used by the **Frame/Axis** and **Plane** object types.

Relationship - Specifies the relationship between the selected orientation object and the assembly. If the object is modified, then the manufacturing orientation of the assembly is automatically updated. If the object is deleted, the manufacturing orientation does not change, but the relationship between the object and the assembly is deleted.

- **No Relation** - The software maintains no relationship between the orientation object and the assembly. The orientation object does not display when you select an existing assembly for modification. If you modify the orientation object, the assembly orientation does not change.
- **Relation with Update** - The software maintains the relationship between the orientation object and the assembly. The orientation object displays when you select an existing assembly for modification. If you modify the orientation object, the assembly orientation updates.
- **Relation with No Update** - The software maintains the relationship between the orientation object and the assembly. The orientation object displays when you select an existing assembly for modification. If you modify the orientation object, the assembly orientation does not update. This option is the default value.



Edit View Direction

Allows you to modify the view direction of the assembly when viewing the assembly in manufacturing orientation and assembly method drawings. The view direction symbol is an arrow placed in a quadrant of the manufacturing orientation plane. A default viewing direction is initially provided based on rules. You can change the viewing direction to be used by selecting a new quadrant. The arrow displays in the new quadrant.

X Rotation

Rotates the manufacturing orientation plane with respect to the local x axis. Type a positive value for clockwise rotation about the axis. Type a negative value for counterclockwise rotation about the axis.

Y Rotation

Rotates the manufacturing orientation plane with respect to the local y axis. Type a positive value for clockwise rotation about the axis. Type a negative value for counterclockwise rotation about the axis.

Z Rotation

Rotates the manufacturing orientation plane with respect to the local z axis. Type a positive value for clockwise rotation about the axis. Type a negative value for counterclockwise rotation about the axis.

Toggle Orientation

Toggles the direction of the local z manufacturing up-vector about the local x-y plane used for definition of the manufacturing orientation plane.

Fit to Assembly

Fits the graphic view to the assembly.

What do you want to do?

- *Create a new assembly or assembly block* (on page 69)
- *Set assembly user-defined planar manufacturing orientation* (on page 70)
- *Set assembly manufacturing orientation by object* (on page 71)
- *Edit an assembly or assembly block* (on page 72)

Create a new assembly or assembly block

1. Click **New (Block) Assembly** .
2. Select content for the new assembly or assembly block from the **Workspace Explorer** or from the model.
3. In the **Type** box, select **Assembly** or **Assembly Block**.

   NOTE For **Assembly**, you can also select **Config Project Root** as the parent assembly.
4. In the **Parent Assembly** box, select a parent assembly or block for the new assembly.
5. On the ribbon, click **Add Secondary Objects** .

   *The **Secondary Objects** dialog box appears.*
6. Select the types of secondary objects to include in the new assembly and click **Close**.
7. If needed, click **Manufacturing Orientation** to specify a manufacturing orientation for the assembly.

   Set assembly user-defined planar manufacturing orientation (on page 70)

   Set assembly manufacturing orientation by object (on page 71)
8. If needed, type a new name for the assembly in the **Name** box.
9. Click **Finish**.

NOTES

- You can also create new assemblies by right-clicking in the **Workspace Explorer** and selecting **New Assembly** from the shortcut menu.
- In the **Workspace Explorer**, you can use drag and drop to change assembly content, or you can use cut and paste.

Set assembly user-defined planar manufacturing orientation

This procedure applies to a new or an existing assembly or assembly block.

1. On the **New (Block) Assembly** ribbon, click **Manufacturing Orientation** .
2. In the **Orientation Method** box, select **User Defined**.
3. Specify the plane for the manufacturing orientation baseplate using one of the following methods:

   Define a coincident plane (on page 25)

   Define an offset plane (on page 26)

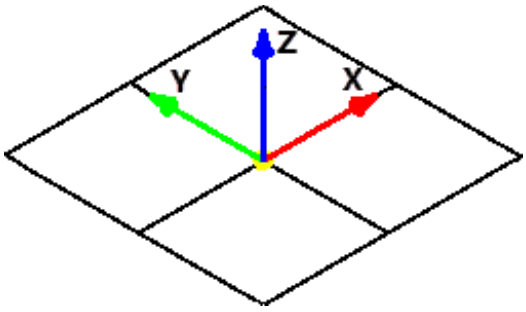
   Define plane using angle from plane (on page 27)

   Define a plane using a point and a normal vector (on page 28)

   Define a plane using three points (on page 29)

   Define plane by vectors normal (on page 29)

   A graphic of the x-y plane and the z up-vector direction displays in the graphic view.



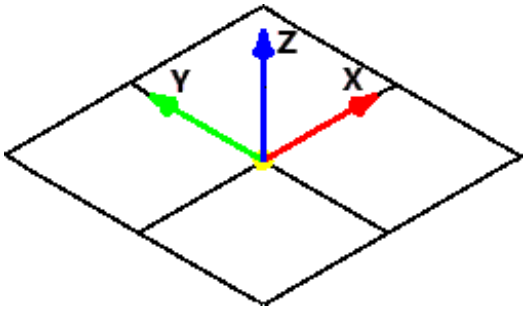
4. If needed, click **Edit manufacturing orientation** to toggle the direction of the z' up-vector.
5. If needed, type values for **X Rotation, Y Rotation,** or **Z Rotation** to rotate the base plate about each coordinate system axis.
6. Click **Finish**.

Set assembly manufacturing orientation by object

This procedure applies to a new or an existing assembly or assembly block.

1. On the **New (Block) Assembly** ribbon, click **Manufacturing Orientation** .
2. In the **Orientation Method** box, select **By Object**.
3. In the **Object Type** box, select **Plate**, **Pin Jig**, **Intersection Seam**, **Seam (By Rule)**, **Coordinate System**, **Assembly**, **Frame/Axis**, or **Plane**.
4. In a graphic view or the **Workspace Explorer**, select an object appropriate to the selected object type:
   - For **Plate**, select a plate part to use as a baseplate. The normal direction on the baseplate is used as the up-vector.
   - For **Pin Jig**, select a pin jig. The base plane of the pin jig is used as the baseplate. The normal direction on the baseplate is used as the up-vector.
   - For **Intersection Seam**, select an intersection seam between a plate and a profile. The direction of the seam is used as the up-vector.
   - For **Seam (By Rule)**, select an intersection seam between a plate and a profile that lead to slots on the plate after detailing, and the slot has a positive opening angle. The direction of the seam is used as the up-vector.
   - For **Coordinate System**, select a coordinate system. The direction of the Z-axis is used as the up-vector.
   - For **Assembly**, select a sub-assembly of the existing assembly. The orientation of the sub-assembly is used for the assembly.
   - For **Frame/Axis**, select an axis of a coordinate system. The direction of the grid plane or axis is used as the up-vector.
   - For **Plane**, select a plane, such as a grid plane, or a planar surface on a plate. The normal direction on the plane is used as the up-vector.

   A graphic of the x-y plane and the z up-vector direction displays in the graphic view.



5. If needed, click **Edit manufacturing orientation** to toggle the direction of the z' up-vector.
6. If needed, type values for **X Rotation, Y Rotation,** or **Z Rotation** to rotate the base plate about each coordinate system axis.
7. Click **Finish**.

Edit an assembly or assembly block

1. Click **Select** on the vertical toolbar.
2. Select **Assemblies** or **Assembly Blocks** in the **Locate Filter**.
3. Select the assemblies or assembly blocks to modify.
4. Using the ribbon controls, edit the assemblies or assembly blocks as needed.

Assembly Properties Dialog Box

Defines the building method properties for the assembly you are viewing.

General Tab (Assembly Properties Dialog Box) (on page 72)
Relationship Tab (on page 34)
Configuration Tab (on page 32)
Notes Tab (on page 33)

General Tab (Assembly Properties Dialog Box)

Displays and defines the general property names and values of the selected assembly or assembly block. Only information that appears in a field with a white background may be edited. All other fields are read-only.

Category

Select to display a group of assembly properties. Available settings are **Standard**, **Planning**, **Weight and CG, Assembly Range, and Position and Orientation.**

Update

Calculates the non-persisted properties, such as length, weight and height.

Standard Category

Name

Name of the assembly. This name is usually generated by the active name rule. You can type a different name for the assembly if you want. Names generated by a rule include a Global Workshare name rule ID, if the name rule ID was defined when the model database was created. For more information, see "Using Global Workshare" in the *Global Workshare Guide*.

Name Rule

Name rule used to generate the name. This field allows you to have optional rules for different situations. The content of the list comes from the "Rule Options" function. Set this option to **User Defined** if you want to specify the name yourself.

Assembly Type

Type of planning assembly, such as block, assembly, or pipe spool. The drop-down list contains all available assembly types as defined in the reference data.

Parent Assembly

Displays the name of the parent assembly, block, or assembly block.

Symmetrically Related Assembly

Displays the name of the symmetrically related assembly if one has been defined using the Mirror Assembly Command . If no relationship is defined, **<None>** is displayed.

Assembly Stage

Construction stage of an assembly from a facility-defined rule set. The drop-down list contains all available stages as defined in the reference data. Examples of stages include part production, assembly of subassemblies, assembly of sections with outfitting, and assembly of sections before painting. If no stage is defined, **<None>** is displayed.

Workcenter

Assigned assembly workcenter from a facility-defined rule set. The drop-down list contains all available workcenters regardless of their level in the workcenter hierarchy. Examples of workcenters include docks, shops, bays, and panel lines. Workcenters may also be areas and zones where a block is assembled from multiple assemblies.

Default Build Equipment

Primary equipment used to join the elements of the assembly. The drop-down list contains all available build equipment as defined in the reference data. Examples of build equipment include jig floors and bending equipment. Build equipment can be changed at the assembly element level by clicking **Sequence Form** on the **Sequence** ribbon. For more information, see *Sequence* (on page 79).

Default Weld Equipment

Weld equipment used within the assigned workcenter when welding weld defined using *Generate Planning Joints* (on page 90). The drop-down list contains all available weld equipment as defined in the reference data.

Exit Equipment

Crane or transport equipment within the assigned workcenter that moves the finished assembly to its next assembly stage and workcenter. The drop-down list contains all available exit equipment as defined in the reference data.

Building method comments

Allows you to enter comments regarding workcenter or work process assignment or any other issue pertinent to the assembly.

Build And Weld As Sequenced

Specifies if you want to weld objects as they are joined in the assembly sequence. Select **True** to weld each object as it is joined, and select **False** to join all objects together before welding.

Orientation Baseplate

Displays the assigned manufacturing orientation baseplate if one has been defined using the **Manufacturing orientation** option. If no baseplate is defined, **<None>** is displayed.

Orientation Coordinate System

Allows you to select the global coordinate system used to define the local manufacturing

orientation coordinate system in the **Manufacturing orientation** option. Default value is the main coordinate system.

Orientation X Rotation

The rotation of the assembly around the X-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation** option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the assembly is no longer oriented by the baseplate.

Orientation Y Rotation

The rotation of the assembly around the Y-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation** option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the assembly is no longer oriented by the baseplate.

Orientation Z Rotation

The rotation of the assembly around the Z-axis of the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation** option), whether based on baseplate or a three-point plane definition. By modifying the rotational values, you can edit the value to any exact orientation relative to the coordinate system. Manually changing the orientation does not affect the selected baseplate, even though the assembly is no longer oriented by the baseplate.

Boardside

Board of the assembly. The drop-down list contains all available values as defined in the reference data, typically **Center**, **Starboard Side**, **Port Side**, and **Both Sides**. Default value is **Center**.

Manufacturing Frame System

Allows you to select the coordinate system used to process structural manufacturing rules for all objects in the assembly. The drop-down list contains all available coordinate systems as defined in the reference data. If no coordinate system is defined, **No Framesystem** is displayed and the local manufacturing orientation coordinate system (defined in the **Manufacturing orientation** option) is used.

Planning Category

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

TIP You can customize slot and collar rules in the Structural Detailing task to use **Build Method** and **Slot Connectivity** properties to determine the slots and collars placed during detailing.

Weight and CG Category

Weight

Displays the weight of all child parts and assemblies of the assembly. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

CoG X

Displays the position of center of gravity along the X-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

CoG Y

Displays the position of center of gravity along the Y-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

CoG Z

Displays the position of center of gravity along the Z-axis in the **CoG Coordinate System**. **Press update to calculate** is initially displayed because the property value is not persisted in the model.

CoG Coordinate System

Allows you to select the coordinate system for calculation of center of gravity (CoG) coordinates. The drop-down list contains all available coordinate systems as defined in the reference data. Default value is the main coordinate system.

Assembly Range

The range box dimensions of an assembly are based on inputs such as the parts selection criteria, the orientation, the assembly shrinkage, and the assembly margin.

Global - Length

Displays the length of the assembly range box in the global orientation.

Global - Width

Displays the width of the assembly range box in the global orientation.

Global - Height

Displays the height of the assembly range box in the global orientation.

Assembly Orientation - Length

Displays the length of the assembly range box in the assembly orientation.

Assembly Orientation - Width

Displays the width of the assembly range box in the assembly orientation.

Assembly Orientation - Height

Displays the height of the assembly range box in the assembly orientation.

Minimum Bound Box - Length

Displays the length of the assembly range box in the tight box orientation.

Minimum Bound Box - Width

Displays the width of the assembly range box in the tight box orientation.

Minimum Bound Box - Height

Displays the height of the assembly range box in the tight box orientation.

Global - Shrinkage - Length

Displays the length of the assembly range box that includes the assembly shrinkage in the global direction.

Global - Shrinkage - Width

Displays the width of the assembly range box that includes the assembly shrinkage in the global direction.

Global - Shrinkage - Height

Displays the height of the assembly range box that includes the assembly shrinkage in the global direction.

Global - Shrinkage and Margin - Length

Displays the length of the assembly range box that includes the assembly shrinkage and margin in the global direction.

Global - Shrinkage and Margin - Width

Displays the width of the assembly range box that includes the assembly shrinkage and margin in the global direction.

Global - Shrinkage and Margin - Height

Displays the height of the assembly range box that includes the assembly shrinkage and margin in the global direction.

NOTE The length, width, and height values are computed after you click **Update**.

SECTION 5

Merge

Merges the content of two or more blocks or assemblies into one block or assembly.

Merging combines the content of two or more blocks or assemblies into one block or assembly.

After you create a number of blocks or assemblies you may need to merge two or more of them. **Merge** allows you to merge blocks into larger blocks.

If you merge blocks, all the defined volumes and elements (blocks, assemblies, and parts) in each block belong to the new block after the merge. If you merge assemblies, all the defined elements (assemblies and atomic parts) in each assembly will belong to the new assembly after the merge.

You can use this command with assemblies or blocks already selected. Because you can only perform the merge when certain rules are followed (see below), the system checks the items you have selected and, if rules are not complied with, the selection set will be ignored and the command starts in its select step. Alternatively, the command starts in the select step with the items already selected, and ready to receive additional selections or to be finished. You can also use the command without any prior selection.

Rules of a Merge

- Only blocks and assemblies can be merged; you cannot merge parts.
- Blocks and assemblies cannot be merged with one another; you can only merge blocks with blocks or assemblies with assemblies.
- Only blocks sharing a common plane can be merged. That is, only neighbor blocks can be merged.
- Assemblies to be merged must be in the same block.
- The block or assembly will be merged in the higher common level.
- The resulting blocks and assemblies must follow the definitions of blocks and assemblies.
- The merge will not affect parts that you have assigned to neighboring blocks.
- You cannot modify the **Merge** command at a later stage. You must use the **Split Block** or **Modify Block** commands.
- When merging blocks and you get to the last one, use the **Flatten an Assembly or Block (on page 210)** command instead of the **Merge** command to combine the blocks.

Merge Ribbon

Select assembly or blocks

Allows you to select blocks or assemblies to be included in a merge.

Finish

Accepts the assemblies or blocks selected for the merge.

Cancel

Clears the selection set and starts over.

Name

Allows you to type a name for the merged blocks or assembly.

Delete Seam

Select this option to delete the planning seam between the blocks that you are merging. Clear this option to send the planning seam to the **To Do List**. Seams sent to the **To Do List** will have the message: "No intersection for this landing curve. Delete the curve or modify the inputs so an intersection exists." You must redefine the landing curve for the seam to remove it from the **To Do List**.

Merge blocks or assemblies

1. Click **Merge**  .
2. Select the blocks or assemblies to merge.

   TIP You can use the model to select the lowest level of blocks; however, you must use the **Workspace Explorer** to select parent blocks or assemblies.
3. If necessary, type a new name for the merged block or assembly.
4. Click **Finish**.

NOTES

- Only blocks and assemblies can be merged; you cannot merge atomic parts.
- Blocks and assemblies cannot be merged with one another; you can only merge blocks with blocks or assemblies with assemblies.
- Only blocks sharing a common plane can be merged. That is, only neighbor blocks can be merged.
- Assemblies to be merged must be in the same block.
- The block or assembly will be merged in the higher common level.
- The resulting blocks and assemblies must follow the definitions of blocks and assemblies.
- The merge will not affect atomic parts that you have assigned to neighboring blocks.
- You cannot modify the **Merge** command at a later stage. You must use the **Split Block** or **Modify Block** commands.
- When merging blocks and you get to the last one, use the **Flatten an Assembly or Block (on page 210)** command instead of the **Merge** command to combine the blocks.

SECTION 6

Sequence

 Defines the details of the sequence in which individual blocks, assemblies, assembly blocks, and parts of the model structure are to be built.

In the Planning task, you work with decomposing the model structure into assemblies designed for a specific workcenter. The task of designing assemblies and building methods includes deciding the sequence in which individual blocks and assemblies should be built. You make some of these decisions when you design the structure of the assembly hierarchy.

An assembly sequence determines the sequence of the items within a single assembly or block. You use the resulting information as a resource for determining and documenting building methods. Later in the process, you can use the information for the generation of building method illustrations and animations. With the **Tools > Show Assembly Method** command, you can use assembly sequence information to show a step-by-step animation of the assembly method for a selected block or assembly. For more information, see *Show Assembly Sequence* (on page 150).

Sequence Ribbon

 **Assembly Properties**

Activates the **Assembly Properties** dialog box. You can use this dialog box to review and modify properties such as assembly type or default build equipment, which you cannot set on the ribbon. For more information, see *Assembly Properties Dialog Box* (on page 72).

 **Select assembly or block to be sequenced**

Allows you to select the assembly, block, or assembly block to be sequenced.

 **Assign sequence**

Assigns an assembly sequence to the content of the assembly or block. The items contained in the selected assembly or block are sequenced by the order in which they are selected.

 **Sequence Form**

Opens the **Assembly Sequence Form** dialog box, which allows you to create, review, and modify the assembly sequence. For more information, see *Assembly Sequence Form Dialog Box* (on page 80).

Finish

Commits the assigned assembly sequence to the database.

 **Cancel**

Clears the currently selected blocks and/or assemblies and starts over.

Define the assembly sequence

1. Click **Sequence** on the vertical toolbar.
2. In the **Workspace Explorer**, select the assembly, block, or assembly block that you want to sequence.
3. Click **Assign Sequence** on the ribbon, and select the first object for the sequence.
4. Continue to build the assembly sequence by selecting objects in the order needed for the sequence.
5. Click **Finish**.

NOTES

- You cannot select assemblies or assembly blocks graphically; you must select them from the **Workspace Explorer**.
- If you use the **Sequence** command with some or all items already sequenced, the software adds newly sequenced items to the end of the previously established sequence.
- You can review and modify the assembly sequence on the **Sequence** ribbon with **Sequence Form**.

Assembly Sequence Form Dialog Box

Allows you to create, review, and modify the assembly sequence. The sequence determines the order of assembly of the objects (parts, assemblies, assembly blocks) within an assembly or assembly block.

TIP Sequences can also be created using the **Sequence** command. For more information, see *Sequence* (on page 79).

Index

Specifies the assembly order of the selected object.

Name

Lists assembly objects, which are all planning objects that are a first-order child of the selected assembly, excluding planning joints. A sequence number for the objects that have been assigned to the sequence precedes the object name. Unsequenced objects are listed below the sequenced objects and do not have a sequence number. By default, objects are not sequenced. Selecting objects in the list highlights the object in the graphics view and enables the relevant **Move up** and **Move down** buttons.

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build

method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

Build Equipment

Identifies the equipment being used to join the objects of the assembly. The drop-down list contains all available build equipment as defined in the reference data. Examples of build equipment include jig floors and bending equipment. Values from the **Default Build Equipment** property on the *General Tab (Block Properties Dialog Box)* (on page 52) are displayed by default. Build equipment can be changed for each object, overriding the **General** tab value.

Fit to Selection

Fits the selected objects to the graphic view.

Normalize Indices

Removes any gaps that may exist in the index. For example, if the existing index sequence is **1**, **2**, and **4**, **Normalize Indices** changes the sequence to **1**, **2**, and **3**.

Refresh Indices

Sorts the indices in ascending order.

Paste Rows

Inserts previously cut rows above the selected row.

Cut Rows

Copies and removes the selected rows. Click **Paste Rows** to add them back to the table. If you do not click **Paste Rows**, the cut rows stay at their current location.

Move to Top

Assigns the selected object with the first index.

Move Up

Moves the selected objects up one row in the grid. Moving a grouped, sequenced object moves the entire group.

Move Down

Moves the selected object down one row in the grid. Moving a grouped, sequenced object moves the entire group.

NOTE Moving an object up or down does not sequence it unless the object to which it is being moved above or below has been sequenced.

Move to Bottom

Assigns the selected object with the last index.

Sequence/Unsequence

Assigns or removes an assembly sequence to or from the content of the assembly or block. The items contained in the selected assembly or block are sequenced or unsequenced by the order in which they are selected.

NOTE The **Sequence** and **Unsequence** button toggles, based on whether the selected

object has been sequenced. If both sequenced and unsequenced objects are selected, then the button is inactive.

Select All

Selects all of the sequences in the table.

Group

Groups the selected sequenced objects so that they have the same assigned sequence number. Also sequences objects if they were not previously sequenced. The button remains disabled until multiple objects are selected. Defining a new group overrules groups that have been previously defined. Groups are dissolved if unsequencing is applied.

Ungroup

Ungroups the highlighted grouped objects. The button is disabled if no grouped objects are selected.

Apply Rule

Applies the rule selected in the drop-down list to all selected items (sequenced and nonsequenced). The standard rules delivered with the software are **SequenceBySize** (projected area in manufacturing orientation descending) and **SequenceByWeight** (object weight descending). For more information, see "Assembly Sequencing Rules" in *Planning Reference Data*.

Auto Normalize

Automatically normalizes the indices after each operation.

Auto Refresh

Automatically refreshes the indices after each operation.

Auto Highlight

Automatically highlights the selected objects in the graphic view.

SECTION 7

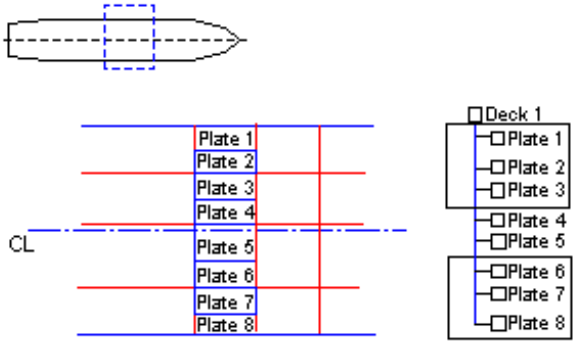
Mirror Assembly

Copies a subset of the assembly hierarchy structure from one side of the model to the other. Before executing the **Mirror Assembly** command, you must first define a source and a target. The software identifies parts at the source location that have similar or symmetric parts at the target location. At the target side, the software creates the necessary source-side assemblies in order to assign the similar or symmetric counterparts.

The **Mirror Assembly** command overrides any conflicting assembly hierarchy assignments that already exist in the target assembly hierarchy area. Additionally, the **Mirror Assembly** command forms relationships between the symmetrically related assemblies and blocks based on your input prior to executing the command.

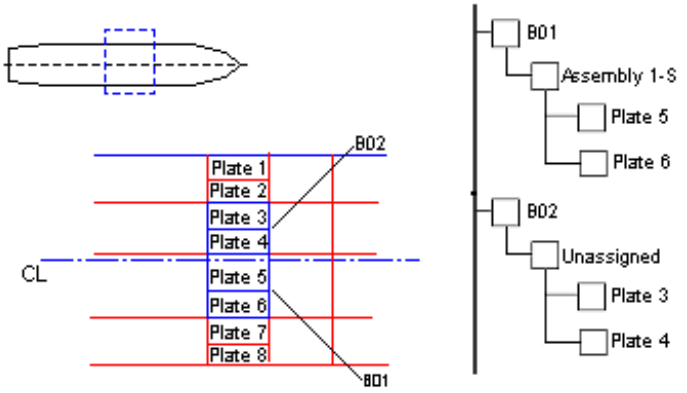
Managing Symmetrical Block Definitions

Parts (structural part items, connection items, edge features, holes, and so forth.) that are defined in the Molded Forms and Structure tasks can exist as part of a group of similar structures, meaning that a part in starboard side can have a counterpart on the port side as shown in the following illustration:



NOTE Plates 4 and 5 are not symmetrical due to the centerline (CL) - Plate 5 is wider than Plate 4.

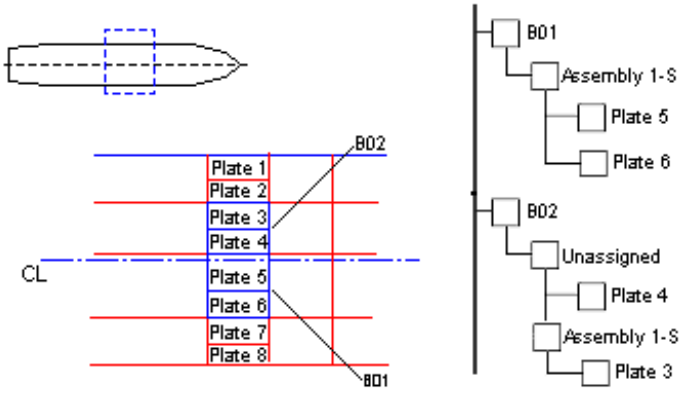
Using the Planning task, the planner creates blocks and assemblies. In the following illustration, an assembly structure that includes Plates 5 and 6 has been defined in block B01 (port side). All the parts in block B02 (starboard side) remain unassigned:



Using **Mirror Assembly**, you can specify that the assembly structure in B02 be the same as that in B01. In so doing, changes to the assembly structure in B01 are reflected in B02 and vice versa. The **Mirror Assembly** command copies part of the assembly hierarchy structure of one block to another and finds symmetric parts in the other block to assign to the assemblies created.

NOTE Symmetrical parts are found based on symmetry relations created when **Tools > Board Management Service** is run from the Structural Detailing task. For more information, see the *Structural Detailing User's Guide* available from the **Help > Printable Guides** command.

Only that part of the assembly structure where symmetric parts are found is copied. If no symmetric parts are found in the other block, no mirroring is done. In the following illustration, plate 4 is not assigned to the assembly because it has no counterparts on the starboard side:



NOTE Unassigned parts (those with no symmetrical counterparts) are assigned to the assembly manually.

Mirror Assembly is a one-way operation. Using the command, however, does not direct the software to continuously listen for new or edited symmetric parts or assemblies. The **Tools > Synchronize Hierarchy** command is the corresponding bi-directional synchronization of unassigned parts in symmetrically related blocks and assemblies. If new parts have been added, the command allows for an update of the assembly hierarchy structure without the loss of non-symmetrical assembly hierarchy deviations made in either side. Synchronization of the

hierarchy builds on the relationships formed between symmetrical assemblies and blocks when using **Mirror Assembly** or when manually set by the planner. For more information, see *Synchronize Hierarchy* (on page 154).

Mirror Assembly Ribbon

Select source node

Select the leaf level block or assembly to be mirrored. A block must have child blocks, assemblies, or parts. An assembly must have child assemblies or parts.

Select target node

Select the leaf level block or assembly to mirror.

Finish

Accepts your input and begins processing.

Mirror Assemblies

Specifies how the children of the selected source assembly/block will be mirrored. Select **With Symmetric Content** to only mirror a source node with symmetrical children in the target node. Select **All** to mirror all children of the source node to the target node.

Assembly Name Rule

Select the naming rule to use for the target assembly or block. For more information, see Naming Rules in *Planning Reference Data*.

Mirror the assembly hierarchy

1. Click **Mirror Assembly** on the vertical toolbar.
2. Select the leaf level block or assembly to mirror.

   TIP Only leaf blocks are selectable.
3. Select the target leaf block or assembly to mirror.

   TIPS

   - If a leaf level block has been selected to be mirrored, only another leaf level block can be selected.
   - You can use the **Create all** or **Make symmetric** buttons on the **Mirror Assembly** ribbon to specify that all source assemblies be created on the target side or only those assemblies with symmetrical content
4. Click **Finish**.

NOTES

- Any user-controlled properties of assemblies, such as workcenter, equipment, or sequences (symmetric parts only) - are inherited by or copied to the newly created assemblies.
- Parts in the target assembly that are not mated with anything from the source side remain as they were previously assigned.

- Modification of assembly properties on either symmetric assembly does not automatically update the other.
- Deleting a symmetrically related assembly or block has no effect on its symmetrically related assembly or block except for removing the relationship.
- The **Mirror Assembly** command is a one-way operation. Using this command does not cause the software to continuously listen for new or edited symmetric parts or assemblies. For information on the corresponding two-way operation, see *Synchronize Hierarchy* (on page 154).

SECTION 8

Auto Assembly

Automatically creates assemblies based on predefined rules. If you select blocks before you activate the **Auto Assembly** command, the software adds those blocks to the blocks selected during command processing. The user-defined rule can either apply to all parts assigned to the block or only to those parts that are in the **UnAssigned Parts** folder. You can change the settings at any time before you run the command.

Creating Rule-based Assemblies

The assembly hierarchy for structural parts includes a multitude of assemblies and sub-assemblies, possibly many thousands per model. The Planning task provides planners with tools that can greatly reduce the time-consuming and repetitive process of creating the required sub-assemblies for structural parts. Based on the content of one or more blocks, **Auto Assembly** imposes rules that, as an output, create new assemblies and assign parts to them. The rules themselves are externalized as Visual Basic reference data; the **Auto Assembly** command merely applies these rules to a user-defined set of blocks.

Rules can be applied to all parts within the block or they can be applied to only those parts that have yet to be assigned to any assembly. Examples of rules to apply per block include the following:

- A profile part and its parent plate is an assembly.
- A bracket with reinforcement is an assembly.
- The collection of parts representing a leaf plate system within a block is an assembly.

Rules in the **Auto Assembly** command are applied to plate parts, profile parts, edge reinforcements, brackets, and collars created in the software, and structural parts imported with the **File** > **Import** > **Ship Structure** command of the Tribon interface. For more information on the Tribon Interface, see the *Tribon User's Guide.*

Auto Assembly Ribbon

Select Blocks

Select the blocks on which you want to apply the rules. Only blocks are selectable. Any blocks that you selected before running **Auto Assembly** are added to the select set that results from this step. Any other selected items are discarded from the select set.

Finish

Executes the command and commits the changes to the database. The software applies the rules to the selected blocks individually. If you select a non leaf-level block, the software applies the rules to that block and excludes any sub-blocks and their content.

Cancel

Discards the select set so that you can start over.

Apply rule

Specifies the automatic assembly creation rule to apply to the select set. The software populates the list with data as defined in the Planning reference data. The following default rules are delivered:

SMAutoAssemblyRule (on page 88)

To

Controls the items on which the software imposes the rules. You can select **All Parts** or **Unprocessed Parts**. For blocks that are not leaf blocks, the software skips any content of block children.

Create a rule-based assembly

1. Click **Auto Assembly** on the vertical toolbar.

   NOTE If you selected blocks at the onset of selecting **Auto Assembly**, the software adds them to the select set chosen in the next step.
2. Select the blocks on which you want to apply the rules.
3. In the **Apply rule** list, select the rule you want to use.
4. In the **To** list, select **All Parts** or **Unprocessed Parts**.
5. Click **Finish**.

NOTES

- You must first run the **Block Assignment** command to place parts in the **UnAssigned Parts** folder for each block. For more information, see *Block Assignment* (on page 156).
- Rules are applied to the selected blocks individually.
- If a non leaf-level block is selected, the rules are applied only to the block itself and exclude any sub-blocks and their content.

SMAutoAssemblyRule

This rule is available as an option for **Apply Rule** when using the **Auto Assembly** command.

Assemblies are created based on:

Surface Area

- A primary plate part, having the largest surface area.
- Other plate parts with the same parent system as the primary plate.
- Plate parts with a manual logical connection to the primary plate.
- Profile parts on the plate parts.
- Edge reinforcements on the plate parts.
- Brackets using the primary plate as a support.
- Brackets using a profile on the primary plate as a support, when the primary plate is a transverse bulkhead.

- Assign child plate parts, profile part, and beam part to the assembly of the plate part.
- Assign a default workcenter and equipment to the assembly.
- Collars for penetrated plates gathered into the assembly. Collars are gathered into subassemblies.

Brackets with Children

A bracket with stiffeners or edge reinforcements is created as a separate assembly.

Bounded Parts

When the primary plate part has no children (that is, connected profiles, edge reinforcements, or brackets), then the software bounded plate parts without children to the assembly. This type of assembly does not apply to a hull or deck primary plate.

The process is repeated for the next plate part not meeting any of the above criteria and having the largest surface area, until all parts are assigned.

NOTE Select **Unprocessed Parts** from the **To** list in the **Auto Assembly** command. If you select **All Parts**, the software processes no parts.

BuiltUpAutoAssemblyRule

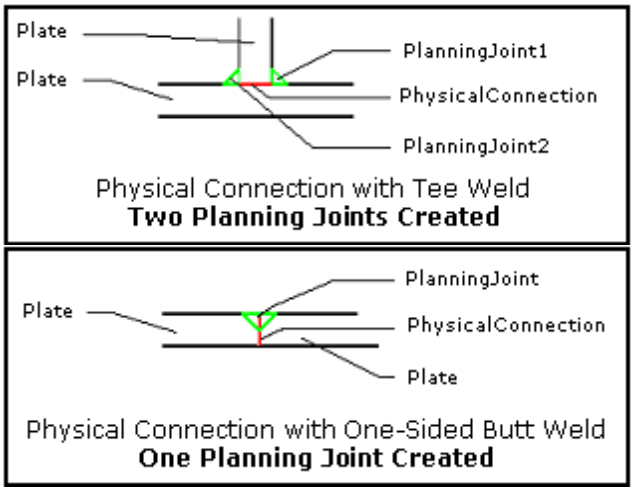
This rule is available as an option for **Apply Rule** when using the **Auto Assembly** command. It creates one assembly for each built up member, with parts added as assembly children. The software updates the assembly created with **Assembly Type** as a built up member, and modifies its orientation using the local coordinate system orientation of the built up member.

SECTION 9

Generate Planning Joints

Creates planning joints for all physical connections in a selected assembly, block, or assembly block.

The planning joint is a decomposition of the physical connection into its manufacturing components or welding tasks for planning purposes, as shown in the following illustrations:



NOTE Parts do not have physical connections until they have been detailed in the Structural Detailing task.

The planning joint is placed on the **Assembly** tab of the **Workspace Explorer** by **Generate Planning Joints**. Planning joints for all welds in an assembly or block are placed in a **Welds** planning joint folder.

The output is one or more planning joints for each physical connection depending on the weld type and position. The parent of a planning joint is a physical connection. However, the planning joint only appears on the **Assembly** tab, while the physical connection only appears on the **System** tab. The initial weld position on which the item is placed is flat. Actual weld positions can only be provided at a later stage (after assembly hierarchy assignment of the planning joint).

Within the Planning task, the planning joint serves the following purposes:

- The ability to plan weld tasks using the block/assembly hierarchy and sequencing. The planning joints are grouped by the part joined to the assembly after assembly sequencing has been performed. The default placement of planning joints is with the assembly or block where the joining parts come together. The planner can, however, move the planning joint to a higher-level assembly or block, and, thus, postpone the welding task.
- The ability to automate bevel selection and attribute definition. Weld position, weld equipment, plate thickness, and material and grade data provide the basis for bevel selection and attribute definition. Weld position and weld equipment data can only be calculated or found through the planning joint and its assembly hierarchy assignment. The

weld position is derived from connection geometry and assembly manufacturing orientation, whereas weld equipment is an assembly attribute.

- The ability to export geometry and data for work content estimation. Work content estimations are based on data such as weld position, fillet length, fillet size, accessibility, and fillet shape, among others.

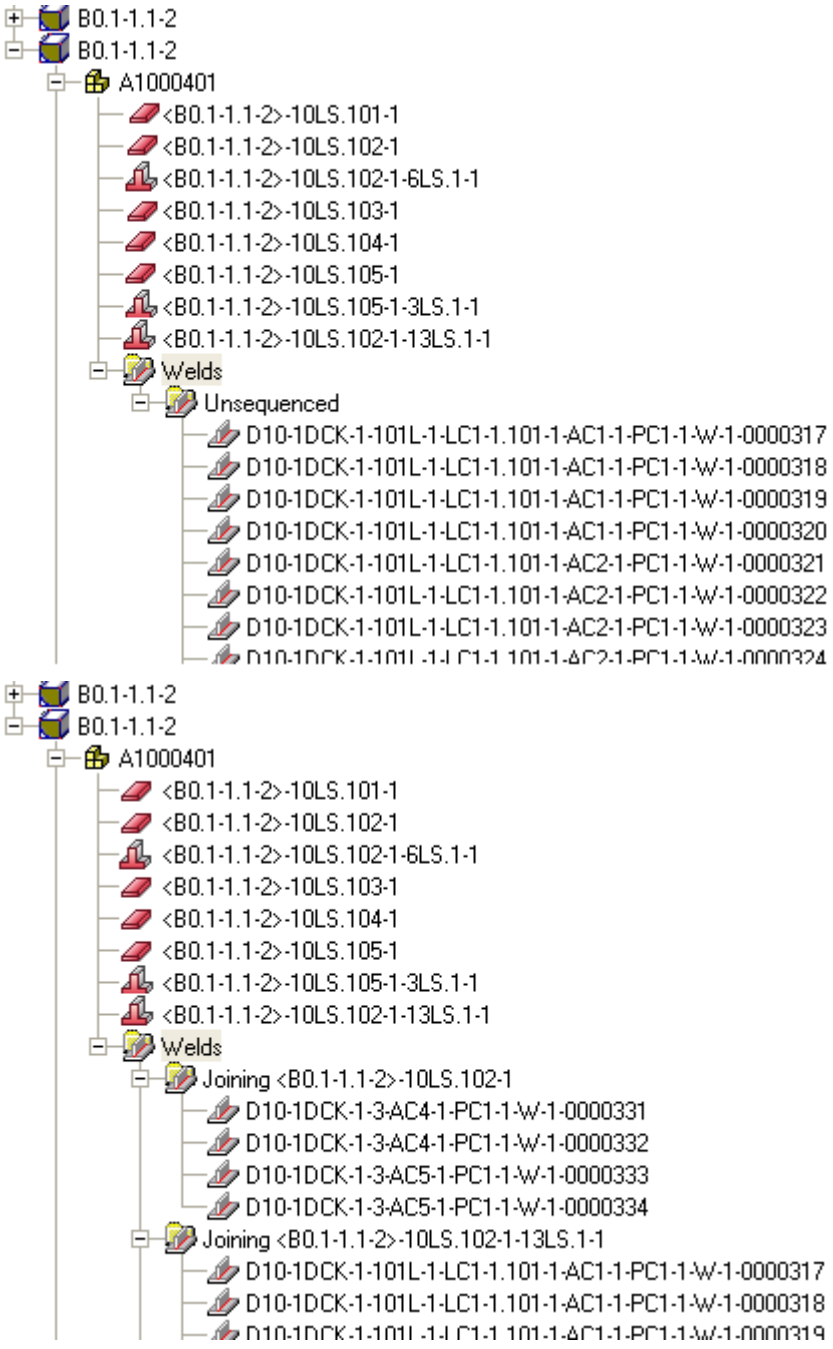
You can view and set property values for the planning joint, including name, name rule, weld equipment (overriding the assembly value), and accessibility.

You can view other properties with values defined by the parent objects of the planning joint. Properties such as inclination and rotation of the planning joint are calculated based on the manufacturing orientation of the assembly where the planning joint welding task is performed. Using inclination and rotation values, the software determines the welding position based on definitions in the Planning reference data. Other properties such as side and shape are defined by the physical connection.

You can split a physical connection in the Structural Detailing task using a split command if, for example, another bevel characteristic is needed. The **Split Planning Joint** command allows you to further split planning joints with identical bevel characteristics (children of the same physical connection) but with individual assembly assignments. For more information, see **Split Physical Connection** in the Structural Detailing task and *Split Planning Joint* (on page 106).

Planning Joint Groups

Planning joint groups are organized in folders and are created based on the assembly and weld sequence, as defined using the **Sequence Form** on the **Sequence** ribbon. For more information, see *Sequence* (on page 79).



The following software-defined groups are available:

- **Welds** - The parent planning joint group. All other planning joint groups are children to the **Welds** group.
- **Unassigned** - Contains planning joints for which at least one part is in the **Unassigned Parts** folder on the **Assembly** tab, as created by the **Block Assignment** Process. For more information, see *Block Assignment* (on page 156).
- **Joining** <assembly name> - Contains planning joints for sequenced assembly objects. The initial (base) object in the assembly sequence does not have a planning joint group because it is not initially welded to anything. The first joining group applies to the second object in the assembly sequence. This group is created when **Build And Weld As Sequenced** is set to **True** on the **General** tab on the parent assembly's properties.
- **Unsequenced** - Contains planning joints for unsequenced assembly objects. An **Unsequenced** group is not created until at least two assembly objects are sequenced, and one joining group is created.
- **Subsequent** - Contains all planning joints for the assembly sequence. This group is created when **Build And Weld As Sequenced** is set to **False** on the **General** tab on the parent assembly's properties.
- **Postponed** - Contains planning joints for parts at a lower assembly level that you have moved to a different assembly.

Custom groups, such as **Stage** and **Module**, are also delivered. Your reference data administrator can modify the custom groups.

Generate Planning Joints Ribbon

Sets options for generating planning joints. The ribbon appears automatically after you click **Generate Planning Joints**  .

Select

Select one or more assemblies, blocks, and/or assembly blocks, either graphically in the active window (blocks only), or through the **Workspace Explorer**.

Finish

Processes the **Generate Planning Joints** command and commits the changes to the database.

Action

Defines what action the command performs.

Create - Creates new planning joints on the selected assemblies.

Update - Updates existing planning joints on the selected assemblies.

Create & Update - Creates new planning joints and updates existing planning joints on the selected assemblies.

Delete - Deletes all planning joints on the selected assemblies.

Planning Joint Rule

Defines the rule from the catalog used to create planning joints.

PlnJointRule Default - Creates planning joints for all edges of the physical connection. The thickness of connected plates determines the planning joints for the side welds of a physical

connection.

PlnJointRule Single Weld - Creates one planning joint for a selected physical connection.

PlnJointRule Weld Category - Creates planning joints based on the weld bevel category. This rule creates one planning joint for a one-sided bevel category physical connection and two planning joints for a two-sided bevel category physical connection.

Submit Job

Displays the **Schedule Planning Joint Generation** dialog box so that you can schedule the creation of the planning joints at some other time. This option is only available if you have selected a block, assembly, or an assembly block. For more information, see *Schedule [Task] Dialog Box* (on page 160).

NOTE Learn more about SmartPlant Batch Services in the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

Edit Planning Joints Ribbon

Sets options for editing planning joints. The ribbon appears automatically after you select a planning joint in the model.

 **Properties**

Displays the **Planning Joint Properties** dialog box, on which you can edit properties such as weld position and weld equipment.

Finish

Commits the modified planning joint to the database.

Parent Assembly

Allows you to change the parent assembly or block. Click **More** to choose another assembly or block with the **Select Assembly** dialog box.

What do you want to do?

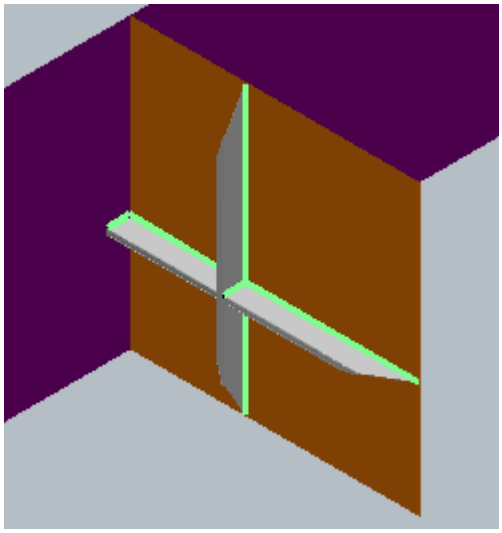
- *Generate a planning joint* (on page 95)
- *Edit a planning joint* (on page 95)
- *Create a new weld group folder* (on page 96)
- *Move a planning joint to a different group folder* (on page 96)
- *Move a planning joint to a different assembly* (on page 97)
- *Schedule a planning joint using Batch Services* (on page 97)

Generate a planning joint

1. Click **Generate Planning Joints** .
2. Select one or more assemblies, blocks, and/or assembly blocks for which planning joints are needed.

   NOTE The assembly must contain detailed plate and profile parts with physical connections. Parts do not have physical connections until they have been detailed in the Structural Detailing task.
3. Select the type of action and the planning joint rule.
4. Click **Finish**.

   The software creates planning joints for each physical connection in the assembly. Planning joints display in green in the graphic view.



NOTE The software places planning joints in a folder for each assembly or block on the **Assembly** tab of the **Workspace Explorer**.

Edit a planning joint

1. Click **Select** on the vertical toolbar.
2. Select **Planning Joints** in the **Locate Filter** box.
3. Select the planning joint to modify.
4. To change the parent assembly of the planning joint, make a new selection in the **Parent Assembly** box.
5. Click **Properties** and modify the planning joint properties as needed.
6. Click **Finish.**

Create a new weld group folder

1. On the **Assembly** tab of the **Workspace Explorer**, right-click an existing block, assembly, assembly block, or **Welds** folder, and select **Create Weld Folder**.

   *The software first creates a **Welds** folder if it does not already exist. The software then creates a custom weld group folder under the **Welds** folder.*

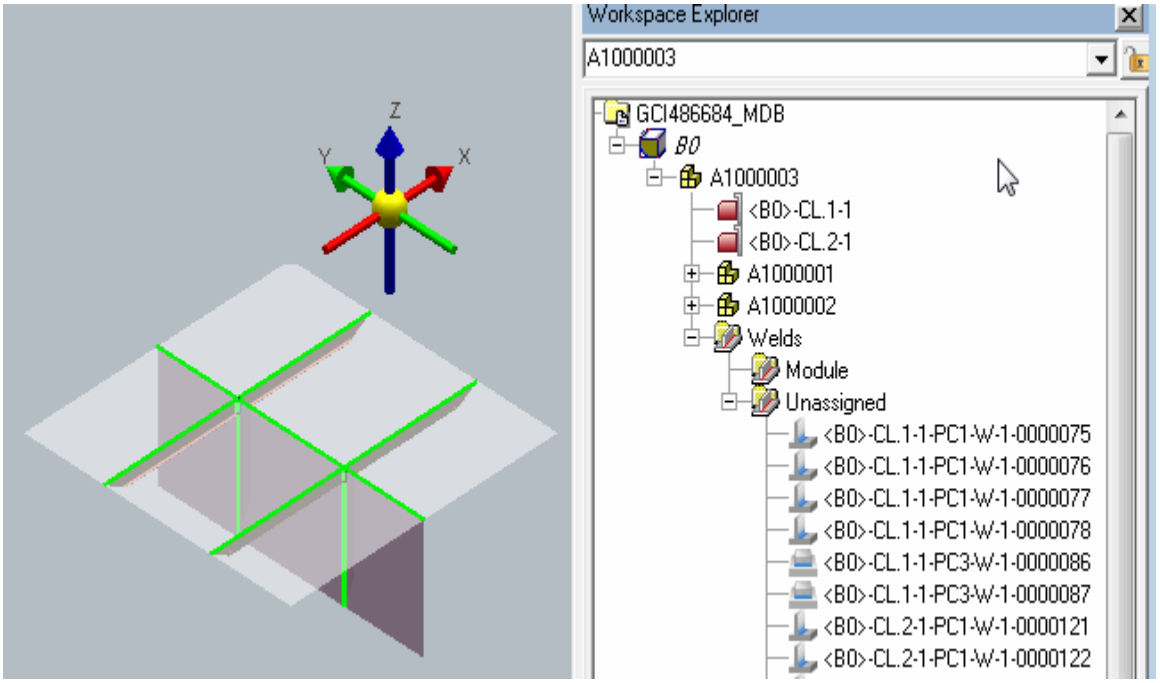
2. To change the name or type of the custom folder, right-click the folder and select **Properties**.

   *The **Planning Joint Folder Properties** dialog box displays.*

3. To change the folder type, select a new custom value, such as **Stage** or **Module**, for **Folder Type**.

4. To change the folder name, select **User Defined** for **Name Rule**, and then type a new value for **Name**.

Move a planning joint to a different group folder



1. On the **Assembly** tab of the **Workspace Explorer**, open a weld group folder containing planning joints.

2. Select one or more planning joints (butt weld, lap weld, or tee weld) and drag the joints to another weld group folder.

   The software moves the planning joints to the new folder.

Move a planning joint to a different assembly

1. On the **Assembly** tab of the **Workspace Explorer**, open a weld group folder containing planning joints.
2. Select one or more planning joints (butt weld, lap weld, or tee weld) and drag the joints to another assembly.

   *The software moves the planning joints to the **Postponed** folder under the new assembly. The software creates **Welds** and **Postponed** folders if required.*

Schedule a planning joint using Batch Services

1. Click **Generate Planning Joints**.
2. Select one or more assemblies, blocks, or assembly blocks for which planning joints are needed.
3. Click **Submit Job**.
4. Complete the **Schedule Planning Joint** dialog box to configure batch processing, and then click **OK**.

   The software displays the message, "This job has been scheduled to run on the chosen server."
5. Click **OK** to dismiss the message, and then click **Finish**.

NOTE For more information about SmartPlant Batch Services, see the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

Planning Joint Properties Dialog Box

Specifies the properties for the planning joint you are editing.

General Tab (Planning Joint Properties Dialog Box) (on page 98)

Relationship Tab (on page 34)

Configuration Tab (on page 32)

Notes Tab (on page 33)

General Tab (Planning Joint Properties Dialog Box)

Displays the properties of the selected planning joint weld. Only information that appears in a field with a white background may be edited. All other fields are read-only.

Category

Displays a category of planning joint properties. Select **Standard**, **Planning**, or **Production**.

Standard Category

Name

Specifies the name of the planning joint. If the active name rule generates the name, the field is read-only. If the name is user-defined, the field is editable.

NOTE Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Name Rule

Specifies the name rule from the catalog used to generate the name. Set this option to **User Defined** if you want to manually specify the name.

Weld Rule

Displays the weld rule used to create the planning joint.

Weld Status

Displays the status of the planning joint with respect to the physical connection of joined parts. If the planning joint reflects the current physical connection, the status is **Up to Date**. If the physical connection geometry has changed, the status is **Out of Date**. If the physical connection and assembly properties have changed, the status is **Out of Date Properties**.

NOTE To update planning joints, select **Update** in the **Action** box of the **Generate Planning Joints** ribbon.

Weld Split Status

Displays the split status of the planning joint. The status options include **ByRule**, **By User**, and **Not Split**.

Root Weld

Indicates whether the weld is a root (**Yes**) or leaf (**No**) weld.

Active Weld

Indicates whether the weld is active or inactive.

Planning Category

Planning properties are populated by the physical connection reference data and are read-only.

By Rule

Specifies the method used to define planning properties. Select **User Defined** to manually define the properties. Select **Rule Based** to populate the properties with the physical connection reference data.

Weld Type

Specifies the type of weld, such as **Tee Weld**, **Butt Weld**, or **Lap Weld**.

Weld Category

Within the selected weld type, specifies the weld category, such as **One Sided Bevel** or **Two Sided Bevel**.

Weld Length

Specifies the design pass length of the weld.

Weld Field Fit Length

Specifies the pass length of the weld based on field conditions.

Weld Thickness

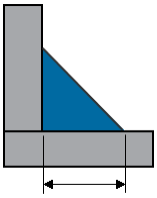
Specifies the overlap thickness of the connected parts. Smart 3D only uses this value for tee and butt welds.

Weld Fillet Measuring Method

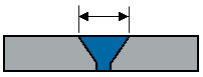
Defines the method of measuring the weld size. Select one of the following values:

- **Leg** - The leg length of the weld.

  Fillet weld:

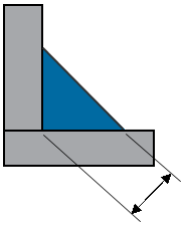
  

  Butt weld:

- **Throat** - The throat thickness of the weld.

  Fillet weld:

  Butt weld:

Weld Fillet Size

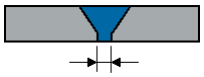
Specifies the weld fillet size based on the **Weld Fillet Measuring Method** value (**Leg** or **Throat**).

Weld Anti Ref Fillet Size

Specifies the weld fillet size on the opposite side of a double-fillet weld based on the **Weld Fillet Measuring Method** value (**Leg** or **Throat**).

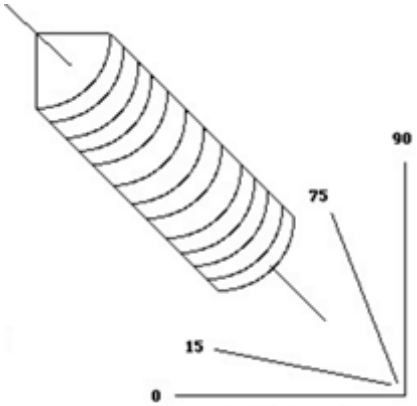
Weld Root Gap

Specifies the gap distance between the parts of the weld.



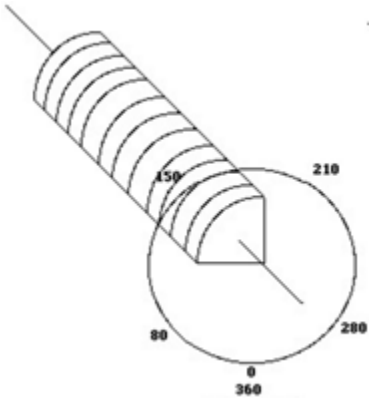
Weld Inclination

Displays the inclination of the axis for the planning joint measured from the horizontal plane. The value is calculated based on the manufacturing orientation of the assembly.



Weld Rotation

Displays the rotation of the weld face about the negative Z axis or vertical axis. This value is calculated based on the manufacturing orientation of the assembly.

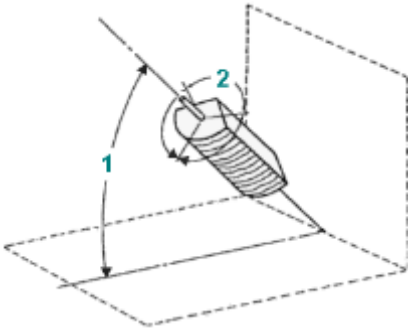


Weld Position

Displays the weld position computed based on:

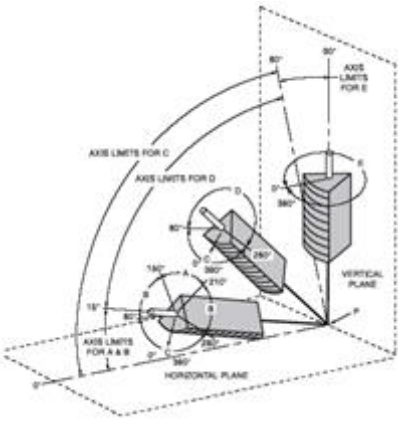
- The degrees of the inclination of the weld axis.
- The degrees of the rotation of the weld face.

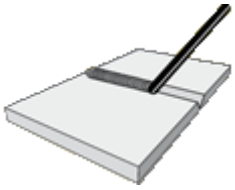
The software calculates this value based on the manufacturing orientation of the assembly. The available values for the weld position are **Flat**, **Horizontal**, **Vertical**, or **Overhead**.



1. Inclination
2. Rotation

Weld positions for BUTT welds:





Flat



Horizontal



Overhead

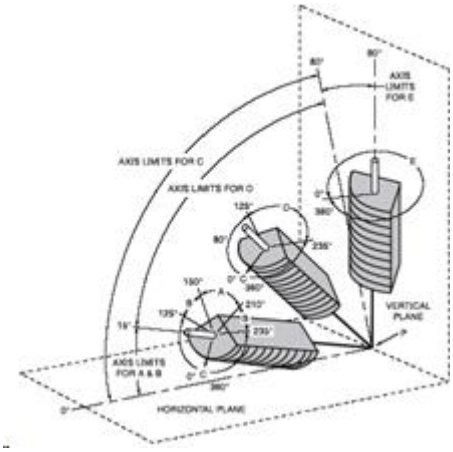


Vertical
Direction = up



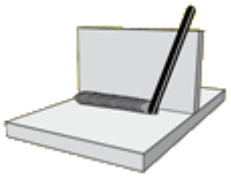
Vertical
Direction = down

Weld positions for fillet welds:

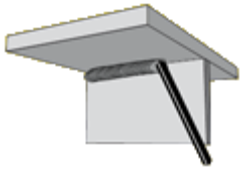




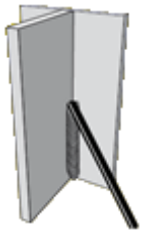
Flat



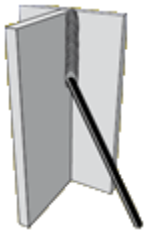
Horizontal



Overhead



Vertical
Direction = up



Vertical
Direction = down

Weld Anti Ref Position

Displays the weld position on the opposite side of a double-fillet weld based on inclination, rotation, and manufacturing orientation of the assembly. Displays **Flat**, **Horizontal**, **Vertical**, or **Overhead**.

Weld Side

Displays the plate side on which the planning joint is located.

Weld Shape

Displays the shape of the planning joint landing curve. The shape influences the work content.

Weld Edge Preparation

Displays the edge preparation, or groove type, of the connected plate parts.

Weld Tail Notes

Specifies notes appearing in the tail of the weld symbol on a drawing.

Production Category

By Rule

Specifies the method used to define production properties. Select **User Defined** to manually define the properties. Select **Rule Based** to populate the properties with the physical connection reference data.

Weld Direction

Specifies the direction of welding, such as **Up**, **Down** for a vertical position, or **N/A** for a horizontal position.

Weld Process

Specifies the shop process used, such as **Shielded Metal Arc**, **Gas Tungsten Arc**, or **Plasma Arc**.

Weld Stage

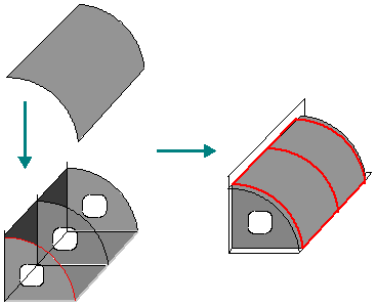
Specifies the construction stage where welding is performed, such as **Block**, **Pre-erection**, or **Erection**.

Weld Assembly Type

Specifies the type of assembly where the welding is performed, such as **Assembly**, **Block**, or **Built-up Profile**.

Weld Accessibility

Specifies the weld access to the planning joint. The options are **Closed space** and **Open space**. Welding performed in a closed space is more expensive. As a result, you classify the weld regarding accessibility, depending on the building method. The red lines in the following graphic show closed-space welds.



WPS Number

Defines the welding procedure specification number.

Weld Classification

Specifies the classification of the weld electrode, such as AWS standard **E7018**.

No of Weld Passes

Specifies the number of passes needed to build the weld to its required size.

Weld Material Name

Specifies the name of the welding consumable.

Weld Material Grade

Defines the grade of the welding consumable.

Weld Testing Type

Specifies the required weld inspection method, such as **Visual Inspection**, **Radiographic Inspection**, or **Liquid Dye Penetrant**.

Weld Sequence Number

Specifies the order of the weld in the welding sequence.

Weld Location

Indicates whether the welding takes place in the field or the shop.

Weld Equipment

Defines the equipment used for welding the planning joint. The list contains all available

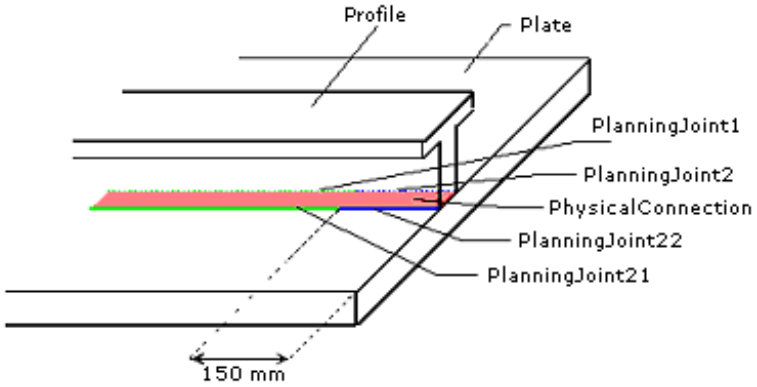
equipment regardless of its level in the workcenter hierarchy, as defined in the reference data.

NOTE If you select equipment that is not assigned to the selected workcenter for the assembly, you receive a warning when an associated manufacturability check rule is executed.

SECTION 10

Split Planning Joint

Splits the planning joint if individual assembly assignments are needed. The following illustrates a planning joint split. Beveling is unchanged on either side, but for fitting purposes 150mm is left unwelded until block assembly.



NOTE The **Split Planning Joint** command cannot change or add bevel characteristics. Use **Split Physical Connection** in the Structural Detailing task if you need to add bevel characteristics.

Split Planning Joint Ribbon

Sets options for splitting planning joints in the model. The ribbon appears automatically after you click **Split Planning Joint**.

Select planning joints to split

Select one or more planning joints to split, either graphically in the active window or through the **Workspace Explorer**.

Select splitting objects

Select one or more reference items to split, either graphically in the active window or through the **Workspace Explorer**. The software checks that the selected items intersect the references.

Method

Select the method of offset from the selected splitting objects to be used to position the split-point location. The direction of the connection, as evaluated by the software, can be one of the following: **None, Longitudinal**, **Transversal**, **Horizontal**, **Vertical**, or **Girth**. If the software determines the major direction to be horizontal, then the **Vertical** option does not display; conversely, if the software determines the major direction to be vertical, then the **Horizontal** option does not display. To locate the global direction, the software projects each connection trace along the main axis.

Offset

Specify an offset distance from the selected splitting objects.

Finish

Processes the **Split Planning Joint** command and commits the changes to the database.

Reject

Cancels the select set.

Accept

Validates the select set.

Split a planning joint

1. Click **Split Planning Joint** .
2. On the **Split Planning Joint** ribbon, click **Select planning joints to split** , and, using the **Workspace Explorer**, select one or more planning joints to split.
3. On the **Split Planning Joint** ribbon, click **Select splitting objects** to select one or more cutting objects to be used as a reference for the split.

   TIP Potential cutting objects include reference planes, other planar objects, and edge features.
4. Define the offset method and type a distance of offset for the split location.

   TIP A preview in a graphical window indicates if an appropriate offset has been chosen.
5. Click **Finish** to commit the split to the database.

NOTE If you delete a split planning joint, all planning joints on that physical connection are also deleted.

SECTION 11

Spooling

Spools are collections of piping parts and the welds that hold them together. The software creates these spool assemblies by applying a set of rules that breaks the assemblies into pieces of a pipeline system. Each piece ends with a spool-breaking component or connection type. After a pipeline, block, or WBS item is spooled, the pipeline, block, or WBS item becomes the parent object of the spools. This process allows spools to be named after the pipeline, block, or WBS item and keeps track of spools that may be out of date, based on the modification date of the parent object.

There are two types of spools. **Standard** spools are based solely upon the pipeline that contains the spools and a set of spooling rules. **Penetration** spools can include parts from one or more pipeline systems. Each of the penetration spools is welded to a common penetration plate.

Generate Spools  and **Create Penetration Spools**  provide a dialog box on which you can select objects for spooling. Each spool must consist of connected objects for the spooling to occur without errors. After generating the spools, the software stores them in the Model database and displays the spools on the **Assembly** tab in the **Workspace Explorer**.

After a spool is generated, the spool can be used to create an isometric drawing for fabrication. These drawings, typically referred to as spool sheets, are tied to the spool and are marked as being out-of-date if the spool is subsequently modified. The spool information can also be included on erection (construction) isometrics. To display the latest spool information on isometric drawings, set the option for showing spool piece identifiers for an isometric drawing style. Then, extract some test drawings using the updated isometric drawing style.

What Spools Include

The software includes in spools any piping component or piece of pipe that is assigned a fabrication type of *by fabricator*. Implied objects (such as bolts, nuts, and gaskets) that are associated with these parts are not included in spools. You can specify options to include an entire pipe support or pipe support parts in the same spool as the components to which they are welded. The supports must have the fabrication requirement set to *by fabricator*, the fabrication type set to *shop fabricated*, and the fabrication responsibility set to *by piping*. These are set in the support properties dialog box under the **Fabrication and Construction** and **Responsibility** categories on the **General** tab.

Additionally, the software provides the ability to create penetration spools, which are used in marine and offshore projects. In penetration spools, multiple piping spools are included in the assembly along with the common penetration plate to which they are attached.

Where Spools Break

When the software processes a pipeline system, spools are created by identifying components or end-preparations that represent intrinsic breaks in the line. The software always breaks spools at field welds, bolted connections, and compression fittings. You can specify whether or not spools break at unions.

In addition, you can place control points at connection objects using the **Insert > Control Point** command to break spools along the pipeline. You have the option to break only at control points,

break at control points and intrinsic breaks, or to ignore control points. For more information, see the *Spool Break by Control Point* information in *Options Tab (Spool Generation Dialog Box)* (on page 112).

If the software finds a gap when processing a pipeline system, the software breaks the spool and proceeds with processing the remainder of the pipeline.

How Spools Are Named

The software uses a default naming rule to give each spool a unique name. The default spool name contains a prefix based on the spool parent and a mark number. The mark number ensures that the spool name is unique and to provide a sequencing of spools within a pipeline. You can modify the naming rules to match your company needs. For more information about naming rules, see the *Smart 3D Reference Data Guide* available from the **Help > Printable Guides** command.

You also can interactively change the name of a spool by selecting it, displaying the **Properties** dialog box, and typing a different name.

Changing the Spooling Basis

You can base your spools on pipelines, planning blocks, or WBS items depending on your needs. If you need to change the spooling basis after spooling has begun, we recommend that you delete any existing spools, change the spooling basis, and then create new spools using the new spooling basis.

To change the spooling basis, in Microsoft Excel, open the *[Product Folder]*\CatalogData\Bulkload\Datafiles\BulkLoadIsoKeys.xls workbook. On the **PipeMfgSpoolRule** sheet in the **SpoolingBasis** column, type **0** to spool by pipelines, **1** to spool by planning blocks, or **2** to spool by WBS items. In column A of that row, be sure to put an **M** to indicate that you have modified the line. Save and exit the workbook. Bulkload the workbook to your catalog using the **Add, modify, or delete records in existing catalog** option.

Specifying Oversized Spools

At creation time, the software checks the size of each spool against the maximum length, height, and width spool dimensions that you can specify. The default values in the reference data are 40 feet by 8 feet by 8 feet, and you can temporarily change these values if required. To change the dimension values permanently, edit options on the **PipeMfgSpoolRule** sheet in the **BulkLoadIsoKeys.xls** workbook and then bulkload the workbook into the catalog.

The software marks spools that exceed the maximum size as oversized so that the designer can insert additional breaks as needed. This check helps to ensure the spools that require dipping in a tank for galvanization, descaling and pickling, and so forth will actually fit in the immersion tank. Also, spools can be marked oversized that are too large to fit in fabrication equipment such as bending machines or annealing ovens, or are too large for shipping. You can check for oversized spools in the following ways:

- View the **Assembly** tab of the **Workspace Explorer**. The icon for an oversized spool contains  .
- Click **File > Define Workspace** and view the **Assembly** tab on the **Filter Properties** dialog box. Again, the icon for an oversized spool contains  .
- View the **Properties** dialog box for the spool. The **SpoolOversizedFlag** property on the **Occurrence** tab is set to **True** if the spool is oversized.

Updating Spools

The software uses a detailed change management routine that prevents existing spools from being arbitrarily renumbered. When a pipeline is re-spooled, only the portion of the pipeline that has changed is impacted by the changes to its spools. New spools are only created as needed and existing spools do not lose their numbers (even if the result is that the numbers are now out of sequence within the pipeline). Existing spools are only deleted as required. This change management ensures that spools are not needlessly modified and that spool sheets are not marked as being out-of-date unless that is absolutely the case.

Deleting Spools

You can delete spools by selecting them on the **Assembly** tab of the **Workspace Explorer** and then selecting **Delete** . The piping parts that were assigned to the deleted spool are reassigned to the deleted spool assembly's parent object. In addition, the software automatically deletes a spool when the last part in the spool is deleted.

Generate Spools

Breaks pipelines into spools. You can view the spools on the **Assembly** tab in the **Workspace Explorer**.

Spool Generation Dialog Box (on page 112)
Spool Properties Dialog Box (on page 116)

What do you want to do?

- *Create spools* (on page 110)
- *Edit spool properties* (on page 111)
- *Delete a spool* (on page 111)

Create spools

1. On the vertical toolbar, click **Generate Spools** .
2. Click the **Spooling** tab.
3. Select the pipelines, planning blocks, or WBS items to spool.

   TIP Use the **WBS Item Type** and **WBS Item Purpose** boxes to aid in the selection of WBS items to spool.
4. Click the **Options** tab.
5. Review the spooling options and edit, if needed.
6. Click **OK**.
7. Click the **Log** tab.
8. Review the spooling log. Click **Save Log** to save the log to a text file, if required.

NOTES

- After spooling, you can view the spools on the **Assembly** tab in the **Workspace Explorer**.

- To show spool identification numbers on an isometric drawing, set **Drawing > Content > Show Spool ID** to **True** in the **Isometric Style Options Browser**.
- You can switch between spooling pipelines, spooling planning blocks, and spooling WBS items. For information on how to switch, see *Spooling* (on page 108).

Edit spool properties

1. On the **Assembly** tab in the **Workspace Explorer**, select a spool, either a standard spool or a penetration spool.

   TIP To create standard piping spools, click **Generate Spools**. To create penetration spools, click **Create Penetration Spools**.

2. Click **Edit > Properties**.
3. Edit the properties as necessary.

NOTE The **SpoolOversizedFlag** property on the **Occurrence** tab is set to **Yes** if the spool is oversized.

Delete a spool

1. On the vertical toolbar, click **Select**  .
2. In the **Locate Filter**, click **Spools**.
3. Click the **Assembly** tab of the **Workspace Explorer**.
4. Select the spool to delete. You can also select the spool in the model.
5. Click **Delete**  .

NOTES

- Parts that were assigned to the deleted spool are automatically reassigned to the deleted spool's parent object in the **Workspace Explorer**.
- The software automatically deletes a spool when the last part in the spool is deleted.
- If you delete an existing spool, sequencing does not reuse the deleted ID.

Spool Generation Dialog Box

Allows you to specify the pipeline systems, planning blocks, or WBS items that you want to spool. You use this dialog box when generating standard spools. If you want to generate penetration spools, click the **Generate Penetration Spools** command.

OK

Processes the pipelines into spools using the options that you specify.

Cancel

Exits the dialog box.

Save Log

Saves the information on the **Log** tab to a text file. For more information, see *Log Tab (Spool Generation Dialog Box)* (on page 115).

Refresh Options from Catalog

Reloads the spool options that are defined in the catalog to the **Options Tab**. For more information, see *Options Tab (Spool Generation Dialog Box)* (on page 112).

Spooling Tab

Allows you to select the objects that you want to spool. For the **Generate Spools** command, you can choose pipeline systems, planning blocks, or WBS items. For the **Generate Penetration Spools** command, you can choose penetration plates.

If you spool by WBS items, two additional drop-down items for **WBS Item Type** and **WBS Item Purpose** appear on the **Spooling** tab. While you are not required to set these options before spooling, the drop-down options provide you with the ability to spool WBS items based on the type and purpose. For example, if you select a **WBS Item Type** of "Design Area" and a **WBS Item Purpose** of "Piping," the WBS items with this particular type and purpose are automatically selected in the tree view, allowing you to spool only these WBS items.

See Also

Generate Spools (on page 110)
Spooling (on page 108)
Delete a Spool (on page 111)
Create Spools (on page 110)

Options Tab (Spool Generation Dialog Box)

Displays the spooling options to use. Each time you open the session file all spooling options default to the spool settings saved in the catalog. You can change the spooling options if needed for a special situation, but you cannot save your changes to the catalog or to the session file. The next time you open your session file, the spool options default to the catalog settings.

If you need to change the spooling options in the catalog, edit options on the **PipeMfgSpoolRule** sheet in the **BulkLoadIsoKeys.xls** workbook and then bulkload the workbook into the catalog.

Name Rule

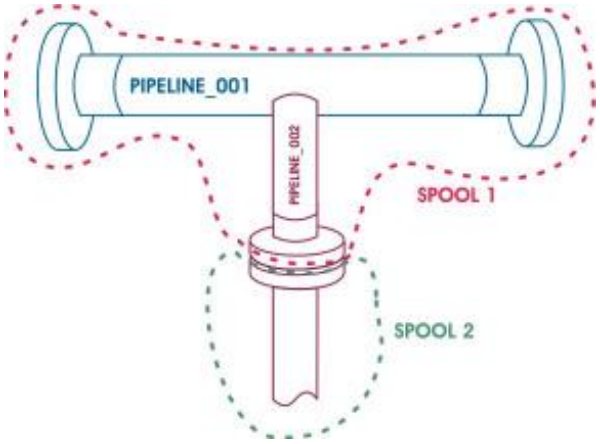
Specifies the naming rule for spools.

Break Spools at Union

Specifies that the software breaks the spools at unions. The union part must have its **Commodity Type** property set to "Union" for the spooling software to recognize the union as a break point.

Include Stub Ins with Header Spool

Specifies that a spool can include the stub-in pipe and all the parts of this stub-in branch until the first spool-breaking component is encountered. For example, by setting this option to **True**, the software spools the stub-in and flange for the isometric drawing with Pipeline_001 even though the stub-in and flange do not belong to Pipeline_001. When creating isometric drawings for Pipeline_001 in the Drawings and Reports task, you must use the **Isometric Style Options Browser** to set the **Drawing.Content.HonorSpoolAtBranch** option to **True**.



Include Items Welded to Pipe in Spools

Includes welded objects, such as pipe hanger or support parts, in the same spool as the components to which they are welded. Surface mounted components are included in the spool if the fabrication type for the surface mounted component is set to "By Fabricator".

Delete Existing Spools

Changes the spool numbering only at modified or added piping. When a drawing is re-extracted, the software recalls the repeated data to avoid changing drawing split points and part, weld, and spool numbers.

This setting guarantees that the software applies the same piece marks (spool names) to spools, even if spools are regenerated.

Ignore Boundaries of Object Being Spooled

When set to **False**, the software will not cross the boundary of the pipeline or block. When set to **True**, the software will cross the boundary of the pipeline or block for spool generation until an intrinsic spool break is found. This feature is intended for use when spooling by block.

To use this option correctly, you need to understand the concept of intrinsic breaks. The spooling software starts a new spool any time it hits an intrinsic break:

- extent of the collection being spooled (pipeline or assembly)
- bolted connection

- (field) welded connection
- change in fabrication category from shop to field
- union (optional controlled by the **Break Spools at Union** option described previously)
- clamped connections

The extent of the collection being spooled is the most important intrinsic break condition. If a pipeline is spooled, the software breaks the spool at the end of the pipeline regardless of what is there. Normally, there is no conflict with the other intrinsic break conditions; however, a conflict can occasionally happen. For example, it does not matter if the pipeline is connected to another pipeline by a shop weld (which would normally not be a spool break), the software will still start a new spool at this point.

In listing the intrinsic breaks in order of importance, the extent of the collection being spooled should always be at the top because it normally trumps any other intrinsic break condition if a conflict arises. Whatever is being spooled, pipeline or block, is the "basket of parts" with which the spool software works. The software does not go outside of that basket for any reason unless the **Ignore Boundaries of Object Being Spooled** is set to **True**. In that case, the spool software steals connected parts from outside of its basket until it hits an intrinsic break, an open end, or it runs out of parts. The spool software will not steal a part from outside the provided basket if that part already belongs to another spool.

TIP In other words, to turn off the first intrinsic break, set **Ignore Boundaries of Object Being Spooled** to **True**.

Maximum Spool Length

Provides the maximum length of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Width

Provides the maximum width of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Height

Provides the maximum height of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

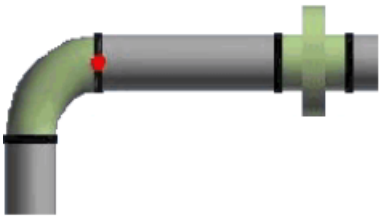
Spool Break By Control Point

Specifies if spools should break at control points.

You can place control points using the **Insert > Control Point** command. You must set the control point **Subtype** to **Spool Break** in order to use the control point with this option. In addition, the control point must be placed on the connection object.

TIP Use **QuickPick** to verify that you have selected the connection object when placing

the control point for the spool break.



Click **Ignore Control Points** to ignore the control points during spooling. Click **Break at Control Points** to break spools at the normal intrinsic line breaks and at control points. Click **Break Only at Control Points** to break spools only at control points.

NOTE The software treats field welds as breaks. If you select **Break Only at Control Points**, the spool also breaks at field welds.

See Also

Generate Spools (on page 110)
Spooling (on page 108)
Delete a Spool (on page 111)
Create Spools (on page 110)

Log Tab (Spool Generation Dialog Box)

Displays a log of the last pipe spooling process. You can save the log to a text file by clicking **Save Log**. The log is divided into four parts:

Before

Lists all the spools that existed in the model before you ran the last spooling process. This is the same list of spools that displays if you select this tab before processing spools.

To Do List

Lists spools that have objects in the **To Do List**. The spools cannot be created until you fix the objects on the **To Do List**. Click **View > To Do List** (or press CTRL+T) to view the **To Do List**.

Modified

Lists all the spools that were modified during the last spooling process.

New

Lists all the spools that were created during the last spooling process.

TIP You can select a spool, pipe, or pipe component in the list to highlight it in the model.

See Also

Generate Spools (on page 110)
Spooling (on page 108)
Delete a Spool (on page 111)
Create Spools (on page 110)

Spool Properties Dialog Box

Displays information about a piping spool. You can create spools using the **Generate Spools** command.

See Also

Occurrence Tab (Spool Properties Dialog Box) (on page 116)
Relationship Tab (on page 34)
Configuration Tab (on page 32)

Occurrence Tab (Spool Properties Dialog Box)

Displays instance-specific information about the spool.

Category

Select the properties that you want to view for the object. Currently, only Standard properties are available for spools.

Standard

Displays all the occurrence properties for the spool as defined in the reference data.

Name

Specifies the name of the spool.

Name Rule

Displays how the object was named. If set to **Default Name Rule**, the software named the object using the default naming rule of <parent pipeline system>_Spool<sequence number>. For example, 1001-P_Spool1 is the first spool of pipeline system 1001-P. If set to **User Defined**, you, or someone else, named the object manually.

Sequence ID

Specifies the object sequence ID.

Spool Oversize Flag

Specifies whether the spool is oversized. The spool size is set on the *Options Tab (Spool Generation Dialog Box)* (on page 112).

True indicates that the spool is oversized.

False indicates that the spool is not oversized.

Create Penetration Spools

Creates penetration spools. This special type of spool consists of one or more spools connected with a penetration plate. There is a tight connection between the spools and the plate part using a penetration plate created in the Hangers and Supports task. You can view the penetration spools on the **Assembly** tab in the **Workspace Explorer**.

IMPORTANT Before you can generate penetration spools, you must have at least one penetration plate in the model. Switch to the Hangers and Supports task and use the **Place Support by Structure** command. For more information about placing penetration plates in the model, see the *Hangers and Supports User's Guide* available from the **Help > Printable Guides** command in the software.

See Also

Penetration Spool Generation Dialog Box (on page 118)
Penetration Spool Properties Dialog Box (on page 120)

Create penetration spools

IMPORTANT Before you can generate penetration spools, you must have at least one penetration plate in the model. Switch to the Hangers and Supports task and use the **Place Support by Structure** command. For more information about placing penetration plates in the model, see the *Hangers and Supports User's Guide* available from the **Help > Printable Guides** command in the software.

1. On the vertical toolbar, click **Create Penetration Spools** .
2. Click the **Spooling** tab.
3. Select a penetration plate for spooling.
4. Click **OK**.

NOTES

- You must have one penetration plate only per pipeline system before penetration spooling can take place. If you have more than one plate connected to the pipe, the software displays an error message.
- After spooling, you can view the spools on the **Assembly** tab in the **Workspace Explorer**.
- To show spool identification numbers on an isometric drawing, set **Drawing > Content > Show Spool ID** to **True** in the **Isometric Style Options Browser**.

Penetration Spool Generation Dialog Box

Allows you to specify the penetration plate that you want to spool. You use this dialog box when generating penetration spools. The dialog box includes the following tabs:

Spooling Tab (on page 112)

Options Tab (Penetration Spool Generation Dialog Box) (on page 118)

Spooling Tab

Allows you to select the objects that you want to spool. For the **Generate Spools** command, you can choose pipeline systems, planning blocks, or WBS items. For the **Generate Penetration Spools** command, you can choose penetration plates.

If you spool by WBS items, two additional drop-down items for **WBS Item Type** and **WBS Item Purpose** appear on the **Spooling** tab. While you are not required to set these options before spooling, the drop-down options provide you with the ability to spool WBS items based on the type and purpose. For example, if you select a **WBS Item Type** of "Design Area" and a **WBS Item Purpose** of "Piping," the WBS items with this particular type and purpose are automatically selected in the tree view, allowing you to spool only these WBS items.

See Also

Generate Spools (on page 110)
Spooling (on page 108)
Delete a Spool (on page 111)
Create Spools (on page 110)

Options Tab (Penetration Spool Generation Dialog Box)

Displays the spooling options to use. Each time you open the session file all spooling options default to the spool settings saved in the catalog. You can change the spooling options if needed for a special situation, but you cannot save your changes to the catalog or to the session file. The next time you open your session file, the spool options default to the catalog settings.

If you need to change the spooling options in the catalog, edit options on the **PipeMfgSpoolRule** sheet in the **BulkLoadIsoKeys.xls** workbook and then bulkload the workbook into the catalog.

Name Rule

Specifies the naming rule for spools.

Break Spools at Union

Specifies that the software breaks the spools at unions. The union part must have its **Commodity Type** property set to "Union" for the spooling software to recognize the union as a break point.

Include Stub Ins with Header Spool

Specifies that a spool can include the stub-in pipe and all the parts of this stub-in branch until the first spool-breaking component is encountered.

Include Items Welded to Pipe in Spools

Includes welded objects, such as pipe hanger or support parts, in the same spool as the

components to which they are welded.

Delete Existing Spools

Changes the spool numbering only at modified or added piping. When a drawing is re-extracted, the software recalls the repeated data to avoid changing drawing split points and part, weld, and spool numbers. This setting guarantees that the software applies the same piece marks (spool names) to spools, even if spools are regenerated.

Ignore Boundaries of Object Being Spooled

When set to **False**, the software will not cross the boundary of the pipeline or block. When set to **True**, the software will cross the boundary of the pipeline or block for spool generation until an intrinsic spool break is found. This feature is intended for use when spooling by block.

To use this option correctly, you need to understand the concept of intrinsic breaks. The spooling software starts a new spool any time it hits one of the following intrinsic breaks:

- extent of the collection being spooled (pipeline or assembly)
- bolted connection
- (field) welded connection
- change in fabrication category from shop to field
- union (optional controlled by the **Break Spools at Union** option described above)
- clamped connections

The "extent of the collection being spooled" is the most important intrinsic break condition. If a pipeline is spooled, the software breaks the spool at the end of the pipeline regardless of what is there. Normally, there is no conflict with the other intrinsic break conditions; however, a conflict can occasionally happen.

For example, it does not matter if the pipeline is connected to another pipeline by a shop weld (which would normally not be a spool break); the software still starts a new spool at this point. In listing the intrinsic breaks in order of importance, the "extent of the collection being spooled" should always be at the top because it normally trumps any other intrinsic break condition if a conflict arises. Whatever is being spooled, pipeline or block, is the "basket of parts" that the spool software works with. The software does not go outside of that basket for any reason unless the **Ignore Boundaries of Object Being Spooled** is set to *True.* In that case, the spool software steals connected parts from outside of its basket until it hits an intrinsic break, an open end, or it runs out of parts. The spool software will not steal a part from outside the provided basket if that part already belongs to another spool.

TIP Setting **Ignore Boundaries of Object Being Spooled** to **True**, turns off the first intrinsic break rule from the previous list.

Maximum Spool Length

Provides the maximum length of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Width

Provides the maximum width of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Maximum Spool Height

Provides the maximum height of a spool for oversizing calculation purposes. You should type **ft** (feet) for the units. If you specify units other than feet, the software uses meters as the units.

Spool Break By Control Point

Specifies if spools should break at control points. You can place control points using the **Insert > Control Point** command. You must insert the control point on the connection. You must also set the control point **Subtype** to **Spool Break** in order to use the control point with this option.

Click **Ignore Control Points** to ignore the control points during spooling. Click **Break at Control Points** to break spools at the normal intrinsic line breaks and at control points. Click **Break Only at Control Points** to break spools only at control points.

See Also

Generate Spools (on page 110)
Spooling (on page 108)
Delete a Spool (on page 111)
Create Spools (on page 110)

Penetration Spool Properties Dialog Box

Displays information about a penetration spool. You create penetration spools using the **Create Penetration Spools** command.

Occurrence Tab (on page 120)
Relationship Tab (on page 34)
Configuration Tab (on page 32)

Occurrence Tab

Displays instance-specific information about the object.

NOTES

- **Import Default Flange Properties**, and **Drive Selection by Generic Port Data** are only available when you are editing the properties of on-the-fly instrument or specialty instrument object.
- If you select multiple objects, then **Import Default Flange Properties** and **Drive Selection by Generic Port Data** are unavailable.

Category

Select the properties that you want to view for the object. Piping object properties are divided into different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, **Bend Data**, and **Port n**.

NOTE **Port n** category is only available when you are editing port properties of on-the-fly instrument or specialty instrument object. If you select multiple objects, then this category is unavailable.

Import Default Flange Properties

Applies default flange port properties for the selected specialty instrument. If you click this

command, the current flange port properties are replaced with corresponding generic port data as defined in the Catalog.

Standard

Displays all the occurrence properties for the part as defined in the reference data. If the table is blank, the object for which you are viewing properties does not have any occurrence properties defined for it in the reference data. Not all standard category properties defined here are available on every part type. For example, the cold spring length property is only available on pipe parts. For more information about occurrences defined in the reference data, see the *Piping Reference Data Guide*.

Name

Specifies the name of the object.

Name Rule

Displays how the object was named. If set to **Default Name Rule**, the software named the object using the default naming rule. If set to **User Defined**, you, or someone else, named the object manually.

Pipe Run

Displays the name of the parent pipe run to which the object belongs. This field is read-only.

Type

Displays the object's short code. The list of available short codes is based on the piping specification and nominal diameter of the object selected along with the geometry at the insertion location.

Option

Displays a list of options that you can use instead of the selected object. If no options exist, this field just displays **Default**. The **Option** list contains any options defined in the piping specification for the short code selected in the **Type** list. For example, if you are looking at a valve part, the **Option** list contains other valve parts, such as one with a higher pressure rating, that you can use here. Selecting a value from the **Option** list updates the object.

Tag

Displays the engineering tag if one is assigned.

Base/Mating Part

Displays whether the selected object is a base part or a mating part. If the object is a mating part, you can use this option to change the object to a base part. Use caution however, because after a mating part has been changed to a base part, it cannot be changed back to a mating part again.

Modeled Length

Specifies the length of the object as it is modeled.

Cut Length

Specifies the cut length of the pipe. Smart 3D calculates the cut length using the following formula:

```
Cut Length = Modeled Length + Field Fit Length End 1 + Field Fit
Length End 2 + Flared Lap Length End 1 + Flared Lap Length End 2 -
```

```
Pipe Bend Elongation - Cold Spring Length + Bend Grip Length + Bend
Pull Length
```

It is possible that the field fit lengths, the flared lap lengths, and the bending elongation, grip, and pull lengths are not applicable to the cut length calculation for a given pipe. In that case, Smart 3D ignores them for that pipe.

Field Liner Thickness

Specifies the field liner thickness if a liner is defined.

Cold Spring Length

Specifies the cold spring length to subtract from the cut length. A positive value makes the **Cut Length** value smaller. The cold spring length refers to intentionally cutting a pipe short and then pulling the assembled pipeline to the connection point. Because of this, there is spring in the pipe until the pipe reaches operating temperature. At that point, there is theoretically no stress on the pipe or the connection.

Reporting Requirement

Select the reporting requirements for the object. You can specify that the object is reported or not reported. An example of an object that you may not want reported is a spacer for a valve. You have to order the valve, but the spacer comes with the valve and does not need to be ordered.

Reporting Type

Select the reporting type. The options that are available here depend on what you select for the **Reporting Requirement**.

Rotation

Specifies the rotation angle of the object.

Stress System Number

Specifies the stress system number for the part.

Isometric Sheet Number

Specifies the isometric sheet number for the part. This property is used in conjunction with **Group Pipe Parts** in Piping.

Sequence ID

Specifies the object sequence ID.

Branch Reporting Ownership Type Basis

Specifies who set the branch reporting ownership. This could be a user or Smart 3D. This option is used by the software when an administrator synchronizes the model with the catalog. The list is defined by the FabricationTypeBasis codelist.

Angle

Displays the angle of the part.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE The **Wet Weight** is the sum of **Dry Weight** and the weight of the fluid inside a piping object. The dry weight and fluid weight values are catalog values entered on the part sheet for the object.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Bend Data

NOTE The Bend Data category is only available if the administrator has bulkloaded the Pipe Bending Manufacturability Rules.xls workbook, and you are viewing properties of a pipe part with bends defined.

Minimum Length Check

Set this option to **True** if you want the software to validate pipe bending manufacturability rules for the selected pipe part.

Bend Type

Defines the pipe bend type.

Preferred Bending Machine

Displays the name of the preferred bending machine as defined in the reference data.

Minimum Tangent Data Set

Select the set of minimum tangent lengths to use with this pipe bend configuration.

End Trimming

Specify whether or not ends are trimmed.

Grip Length

Defines the grip length.

Pull Length

Defines the pull length.

Number of Bends in Piece

Displays the number of bends in the pipe part.

Number of Tangents in Piece

Displays the number of tangents in the pipe part.

Port n

This category is only available when you are editing port properties of an on-the-fly instrument or specialty instrument objects. **n** represents the number of ports of the selected object. For example, when you work with an object that has two ports, then the Category list displays **Port 1** and **Port 2** categories. Select a category to specify the port data for the selected port of that object. The software displays the properties of Port 1 if you select the **Port 1** category, and so forth.

NOTE Use **Import Default Flange Properties** to replace the current flange port properties with corresponding generic port data as defined in the Catalog.

The following properties are logically related and are prioritized in the order they display. **Nominal Diameter n**, **NPD Unit n**, and **Termination Class n** have the highest priority. For example, you can specify a value for **End Preparation 1.** If **Drive Selection by Generic Port Data** is selected, then the software automatically filters and updates the remaining properties with corresponding generic port data as defined in the Catalog.

Port ID n

Specifies a description of the port.

Nominal Diameter n

Specifies the nominal size of the selected port.

NPD n Unit

Specifies the units for the **NPD** value. For example, type **mm** or **in**.

Termination Class n

Specifies the termination class for the part port.

Termination Sub Class n

Specifies the termination sub-class for the port.

End Preparation n

Specifies the end preparation code for the part port. To change the options on the list, edit the **End Preparation** select list in Catalog.

Schedule Practice n

Specifies the schedule practice short description or code for the part port. To change the options on the list, edit the **Schedule Practice** select list in Catalog.

Schedule Thickness n

Specifies the schedule thickness short description or code for the part port. To change the options on the list, edit the **Schedule Thickness** select list in Catalog.

End Practice n

Specifies the end practice for the object. To change the options on the list, edit the **End Standard** select list in Catalog.

End Standard n

Specifies the end standard code for the part port. To change the options on the list, edit the **End Standard** select list in Catalog.

Rating Practice n

Specifies the rating practice for the nozzle.

Pressure Rating n

Specifies the pressure rating code for the part port. Valid codes are listed in the **AllCodeLists.xls** workbook on the **Pressure Rating** sheet in the **Codelist Number** column.

Flow Direction n

Specifies the flow direction code for the port. Specifying the correct flow direction code for each part port is very important because the software automatically orients the part to the flow direction of the pipe or port to which the part is connected. To change the options on the list, edit the **Flow Direction** select list in Catalog.

Piping Point Basis n

Specifies the purpose of the port, such as a vent or a drain, inlet or an outlet, tap by system or by user, jumper or jacket, and so forth. Select an appropriate value from the list.

Drive Selection by Generic Port Data

Filters and lists valid property values as per generic port data defined in the Catalog. This option is selected by default. We recommended that you leave this option selected. If you clear this option, the software does not filter port values and lists all the available values.

You can specify any value. However this can result in errors due to data mismatch. The combination of property values specified might be inappropriate for the port as per the generic port data defined in the Catalog.

SECTION 12

Generate Production Routing

 Generates production data for curved parts. This command allows you to assign production information such as **Workcenter** or **Stage Code**, and assign different routing actions. The data includes different machines used for various actions and their machine codes.

Generate Production Routing Ribbon

Displays the controls used to update and review production routing parts.

 **Select**

Allows you to select an object.

 **Properties**

Starts the **Production Routing Settings** form, which displays **Production Routing** and **Routing Action** properties for each part in a row.

 **Accept**

Accepts the current selection.

 **Cancel**

Clears the current selection.

Production Routing Settings Form

Displays **Production Routing** and **Routing Action** properties for each part in a row.

Name

Name of the part.

Type

Type of system

Status

Status of the part.

Rule

Predefined rule set, user-defined rules are also available. The predefined rule selected from the drop-down will be applicable to all the selected parts.

Workcenter

Place where various production processes are carried out.

Stage Code

Code that indicates the respective work center.

Action

Name of the action.

Machine

Type of machine used to perform the required operation

Code

Displays the machine code.

Apply Rule

Applies the predefined rule to all the selected parts.

Ok

Updates the rules applied.

Cancel

Exits the dialog box.

Delete

Deletes the selected parts.

Fit

Fits all visible or selected objects in the active view.

Routing Property Tab

Displays and defines the production routing data and properties of a part. This tab is available for plate and profile part property page.

Standard Category

Stage Code - Displays the **Stage Code**.

Workcenter - Displays the **Workcenter**.

Rule - Displays the default **Production Routing** rule.

No. Of Actions - Displays the number of **Actions**.

Action - Displays the name of the **Action**. On clicking the **Action**, you can use the **<Insert>**, **<Delete>**, **<Shift Up>**, or **<Shift Down>** to insert, delete, move up, or move down an **Action** respectively**.**

Machine - Displays the machine name.

Code - Displays the code.

Apply rule for production routing

1. Click **Production Routing Settings** on the ribbon.
2. Select rows corresponding to the parts.
3. Click **Apply Rule**.

TIP In the **Action** column, you can insert, delete, move right, or move left an action by using **<Insert>**, **<Delete>**, **<Shift Right>**, or **<Shift Left>** respectively.

SECTION 13

Create Common Part Managers

Processes objects and creates common part managers. The purpose of this command is to create common part managers without having to use the **Common Part Service.**

Create Common Part Managers Ribbon

Properties

Activates the **Properties** dialog box. For more information, see *Common Part Manager Properties Dialog Box* (on page 168).

Select Assembly

Allows you to select the assemblies.

Cancel

Cancels the selected assemblies.

Accept

Accepts the selected assemblies.

Create Common Part Managers

Allows you to select the **Common Part Definition**. This command is activated after you select assemblies and click **Accept** .

Finish

Executes the command and commits the changes to the database.

Name

Displays the name of the common part managers.

Manager Settings Dialog Box

Common Part Definition

Common Part Definition

Displays the common part definitions to be selected.

View Settings

Activates the **View Settings** dialog box.

Manage Filters

Show Existing Managers

Displays the existing managers.

Add Rows

Allows you to add rows.

OK

Closes the **Manager Settings**.

Cancel

Cancels the **Manager Settings**.

Create common part managers

1. Click **Create Common Part Managers** .
2. Click **Select Assemblies** to select the assemblies.

   NOTE You can also select the assemblies as the first step.
3. Click **Accept** .

   *The **Manager Settings** dialog box displays the first **Common Part Definition** in the catalog if no other definition was previously used or selected.*
4. In the **Common Part Definition** combo box, click **More..** if you want to change the **Common Part Definition**.
5. Select the object types and click **OK**.

   *The **Manager Settings** dialog box is closed.*
6. Click **Finish**.

   *The common part managers are created under the assemblies in the **Workspace Explorer**.*

SECTION 14

Common Part Group

Common Part Group creates common part groups. The common parts groups are also created as an entity during **Common Part Service** and is not displayed in the **Workspace Explorer.** You can navigate to **Common Part Groups** through related objects.

Each part that is identified as common is given a **Common Part** name based on a **Naming Rule**. The service does not change the existing name of the part. The parts are displayed as groups under folders of the relevant **Block**, **Assembly** or **AssemblyBlock**. All groups are further grouped according to the type of objects called the **Object Types**.

Common Part Group Ribbon

Displays the controls used to update and review **Common Part Groups**.

Properties

Displays the controls used to update and review the **Common Part Group** properties.

Select

Allows you to select common parts.

Master Part

Sets the selected common part as the master part.

Finish

Accepts your input and begins processing.

Reject

Clears the current selection.

Accept

Accepts the current selection

Assembly

Allows you to select an assembly.

Common Group Type

Allows you to select the object type such as plates or profiles.

Name

Allows you to name the common part group manually, if you do not specify a name, the software uses the default naming rule.

What do you want to do

- *Create a common part group* (on page 133)
- *Select a common part group using filters* (on page 133)

Create a common part group

1. Click **Common Part Group** on the vertical toolbar.
2. In the **Assembly** list, select an assembly.
3. In the **Common Group Type** list, select the group object type.
4. Select the common parts.
5. Click **Accept**.

   By default, the first common part is set as the master part.
6. Click **Reject** to change the selection. Otherwise go to step 9.
7. Select the required common part as the master part.
8. Click **Accept**.
9. Click **Finish**.

NOTES

- You can name the common part group manually by typing a name in **Name**.
- You can create common part groups by selecting a master part using the **Master Part** smartstep command.

Select a common part group using filters

1. Click **Select**.
2. Click **Common Part Group**.
3. Select common parts in the **Common Part Group** or in the related assembly.

NOTE You can select multiple **Common Part Groups** through a dialog box that displays for assemblies consisting of more than one **Common Part Group**.

Common Part Group Properties Dialog Box

Specifies the properties for common part groups.

See Also

General Tab (on page 134)
Relationship Tab (on page 34)
Configuration Tab (on page 32)
Notes Tab (on page 33)

General Tab

Displays and defines the general property names and values of the selected **Common Part Group**.

Standard Category

Name

Displays the name of the group.

Name Rule

Name rule used to generate the name. This field allows you to have optional rules for different situations. The content of the list comes from the "Rule Options" function. Set this option to **User Defined** if you want to specify the name yourself.

Common Parts

Displays the number of common parts

Manufactured Parts

Displays the number of manufactured parts.

Common Part Locked

Displays if the group is frozen or not.

Process Control

Displays if the process is automatic or manual, it can be turned to manual for manual group creation.

Process Purpose

This is meant for future requirement, groups that can be used for other purposes apart from common parts.

SECTION 15

Manufacturing Service Manager

Manufacturing Service Manager (in the Structural Manufacturing task) or **Tools** > **Manufacturing Service Manager** reviews and updates manufacturing objects. **Review** displays a report of the current condition of the manufacturing objects that you have selected. **Updating** pulls any outstanding changes into the manufacturing environment. The command is available on the vertical toolbar in the Structural Manufacturing task, and by clicking **Tools** > **Manufacturing Service Manager** in the Molded Forms, Structural Detailing, Planning, and Structural Manufacturing tasks.

You can select individual parts, assemblies, or blocks for this command.

The **Manufacturing Service Manager** pulls changes into the manufacturing environment instead of associatively pushing from other tasks into the manufacturing environment. In this way, the manufacturing environment is similar to the drawing environment; you have control over which objects get updated and when.

NOTES

- A manufacturing object goes on the **To Do List** for the following reasons:
  - The object has a nonworking status, or it contains objects which are included in different permission groups. You must change the manufacturing part status back to **Working**, and then update the manufacturing part by using the **Manufacturing Service Manager**.
  - The parent system of the manufacturing object was split in the Molded Forms task, and that split migration produced an error.
  - A compute found a failure within the rule and the manufacturing plate, manufacturing profile, or manufacturing member returned a warning.
  - Some other compute failure, with a specific action per entity, occurred.
- The same behavior occurs when the molded form split is performed by someone who does not have write permissions to the manufacturing part. In this case, a second person with write permission to the manufacturing part must update the manufacturing part by the **Manufacturing Service Manager**.

Manufacturing Service Manager Ribbon

Displays the controls used to update and review manufacturing objects.

Selection of Possible Report Data

Opens a dialog box from which you can set up reports.

Select

Selects objects in the model.

Finish

Starts the **Manufacturing Service Manager**.

Cancel

Clears the current selection.

Accept

Accepts the current selection.

Action

Determines whether the **Manufacturing Service Manager** updates the objects, reviews the objects, or reports the Detailed Review of the objects.

Filter

Specify the type of manufacturing part to use.

- **Parts** - All plate, profile, and member parts.
- **Parts or Assemblies** - All plate, profile, member, and different assembly types. For assemblies, you can select blocks, assembly blocks and assemblies.
- **Plate Parts** - Plate parts.
- **Plate Parts and Connected Profiles** - Includes all profiles connected to the plate parts.
- **Profile Parts** - Profile parts.
- **Member Parts** - Member parts.
- **Systems** - All plate systems, profile systems, and member systems.
- **Systems or Assemblies** - All systems and assemblies.
- **Plate Systems** - All plate systems.
- **Plate Systems and Connected Profiles** - All member systems.
- **Profile Systems** - All profile systems.
- **Member Systems** - All member systems and design member systems.

Connected

Specify which type of connected parts to use.

- **<Ignore>** - Only the selected part.
- **Direct Connected** - Includes objects that are physically connected to the selected part.
- **Indirect Connected** - Includes direct objects and objects that are physically connected to the direct objects.

What do you want to do?

- *Review a manufacturing object* (on page 137)
- *Update a manufacturing object* (on page 137)
- *Review or update a set of objects selected by a filter* (on page 137)
- Detail Review a manufacturing object

Review a manufacturing object

1. Click **Select** on the vertical toolbar.
2. Select the manufacturing objects to review.

   TIP You can select the objects in the model or from the **Workspace Explorer**.
3. Click **Review** in the **Action** box.
4. Select the type of manufacturing object to review in the **Report Per** box.
5. Click **Finish**.

Update a manufacturing object

1. Click **Select** on the vertical toolbar.
2. Select the manufacturing objects to review.

   TIP You can select the objects in the model, or from the **Workspace Explorer**.
3. Click **Update** in the **Action** box.
4. Select the type of manufacturing object to review in the **Report Per** box.
5. Click **Finish**.

TIP The software updates the database as each manufacturing object is updated. Clicking **Cancel** stops the update for any objects that have not yet been posted to the database.

Review or update a set of objects selected by a filter

1. Click **More** from the filter selection list.

   *The **Select Filter** dialog box displays.*
2. Set up the filter to meet your requirements. For example, you could define a filter to select plate and profile parts in a specific block.

   The software finds and highlights the objects that match the filter.
3. Click **Service Manager** (in the Structural Manufacturing task) or **Tools** > **Manufacturing Service Manager**.

   The software prompts you to use the selected list of objects.
4. Click **Yes**.
5. If required, add additional objects to the list.
6. Click **Review** or **Update** in the **Action** box.
7. Select the type of manufacturing object to review in the **Report Per** box.
8. Click **Finish**.

Review Dialog Box

Displays the controls used to review the current manufacturing status of manufacturing objects.

NOTE This dialog box opens when you click **Finish** on the **Manufacturing Service Manager Ribbon**.

The manufacturing object status display includes:

- **Up to Date** - Indicates that the object does not require updating.
- **Out of Date** - Indicates that the object requires updating.
- **Can Be Deleted** - Indicates that the object is no longer needed.
- **To-Do-List (Error)** - Indicates that the object is on the **To Do List** because of an error condition.
- **To-Do-List (Out-of-Date)** - Indicates that the object is on the **To Do List** because it requires updating.
- **To-Do-List (Warning)** - Indicates that the object is on the **To Do List** because of a warning condition.
- **Edited** - Indicates that the object has been edited with the part editor. If you modify this part by using the **Manufacturing Plate** or **Manufacturing Profile** commands, or use the **BO Recompute** command, then the status does not display as **Edited**.

Properties

Displays the **Selection of Possible Report Data** dialog box. For more information, see *Selection of Possible Report Data Dialog Box* (on page 139).

Report Per

Displays the current settings for the report data that is set to display. The information that displays depends on what you have set to show or hide on the **Selection of Possible Report Data** dialog box.

Update

Updates the objects. Only the objects shown on the dialog box are updated. This option does not update parts with a status of **Edited**.

Cancel

Closes the dialog box.

File name

Type the path and file name for the report file. You can use the **Browse** button beside the box to browse through the file system.

View

Displays the report file in the default text editor, such as **Notepad**.

Selection of Possible Report Data Dialog Box

Displays the controls used to determine which report data displays on the **Review** dialog box.

Attributes Tab

Option

Determines the manufacturing object type for which you are setting the review options. Select the box next to the option, and change the setting to **Show** or **Hide** as necessary.

Settings Tab

Review Settings

Turn on the settings to apply when the **Action** is set to **Review**. The available options include:

- Only include Out of Date manufacturing objects
- Only include manufacturing objects in **To Do List**
- Check detailed part in **To Do List**
- Include light parts
- Refresh after Update
- Highlight Out of Date objects

Update Settings

Turn on the settings to apply when the **Action** is set to **Update**. The available options include:

- Delete objects under light part
- Update objects with detailed part in **To Do List**
- Update manufacturing objects in **To Do List**
- Recompute all manufacturing objects

SECTION 16

Submit Batch Job

Tools > **Submit Batch Job** runs batch processes using the Intergraph Batch Services framework.

This command runs the following types of batch jobs:

Detailing:

- Detail parts
- Undetail parts
- Update reports

Production:

- Generate manufacturing parts
- Generate XML output files
- Manufacturing Service Manager update

Custom batch processes:

- BO recompute
- Other custom batch processes that you define.

What do you want to do?

- *Run a detailing batch process* (on page 141)
- *Run a production batch process* (on page 141)
- *Run a custom batch process* (on page 142)
- *Add a custom batch process* (on page 142)

Run a detailing batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Detailing** from the **Batch Process Type** list.

   *Smart 3D displays the available detailing batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.

6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Run a production batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Production** from the **Batch Process Type** list.

   *Smart 3D displays the available production batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.

6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Run a custom batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Custom Batch PRocesses** from the **Batch Process Type** list.

   *Smart 3D displays the available custom batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.

6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Add a custom batch process

1. Add the ProgID and job description of the custom batch process to the <Product Folder>\CommonShip\SOM\Client\Xml\CustomBatchJobDetailsConfiguration.xml file.

   For example:

```
<JOB>
   <JobType>CustomBatchProcessD</JobType>
   <JobType_LocaleID>103</JobType_LocaleID>
   <JobDescription>Description of
CustomBatchProcessD</JobDescription>
   <JobDescription_LocaleID>104</JobDescription_LocaleID>
   <JobProgId> ProgID of CustomBatchProcess </JobProgId>
   <IsModelSpecific>False</IsModelSpecific>
</JOB>
```

2. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

3. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

4. Select **Custom Batch Process** from the **Batch Process Type** list.

   *The available custom batch processes display in the **Select Batch Process** list.*

5. Select the custom process to run, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

6. Specify the **Batch Input Type**, and add inputs as necessary.
7. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

8. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Generic Batch Command Dialog Box

Displays the controls used to submit a generic batch command.

Batch Process

Specifies the batch process to run. The list contains recently-selected job types. Click **More** to display the **Batch Process Selection** dialog box, and select a process that is not on the list. For more information, see *Batch Process Selection Dialog Box* (on page 144).

Batch Input Type

Indicates the input type for the batch process.

- **Filter** indicates that you are providing a filter as an input to the batch job.
- **Assembly** indicates that you are providing an assembly as input to the batch job.
- **Object** indicates that you are providing a set of objects as input to the batch job.

 **Add**

Adds inputs to your batch job.

- If **Filter** is selected, this button displays the **Select Filter** dialog box so that you can add filters as inputs to your batch job.
- If **Assembly** is selected, this button displays the **Select Assembly** dialog box so that you can add assemblies as inputs to your batch job.
- If **Object** is selected, this button is not available. To add objects as inputs, select them from the model or the **Workspace Explorer**.

 **Remove**

Removes the selected inputs from the list. The software only removes the inputs that have the associated check box selected.

Share

Saves the inputs for the next batch process. If this check box is cleared, Smart 3D clears the inputs when you select a different batch process.

Schedule as Multiple Jobs

Processes each item in the inputs list as a separate batch process. If this check box is cleared, Smart 3D processes all of the inputs as a single batch process.

Schedule

Displays the **Schedule [Task]** dialog box so that you can schedule the batch process. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Cancel

Exits the command.

Batch Process Selection Dialog Box

Displays the controls used to specify the type of batch process to run.

Batch Process Type

Displays the general batch process types. Select a type to display the available batch process of that type in the **Select Batch Process** list.

Select Batch Process

Displays the available batch process associated with the selected type.

SECTION 17

Un-Split

The **Tools** > **Un-Split** command merges previously split plates into a single plate. The splitting seam remains in the model so that you can use the seam to split the plate again at a later time.

Un-Split Ribbon

Displays the options that you use to remove a split.



Seam Properties

Opens the **Seam Properties** dialog box, which you use to view and modify the properties of the seam that you are about to un-split in the model. For more information, see *Seam Properties Dialog Box* (on page 145).



Select

Specifies the seam that caused the split that you want to un-spit.

Un-Split

Removes the split from plates affected by the selected seam, merges the plates back into a single plate, and leaves the selected seam in the model.

Un-split a plate at a selected seam

1. Click **Tools > Un-Split** from the menu.
2. Click **Select**  on the ribbon.
3. Select the seam that caused the plate split to undo.
4. Click **Un-Split**.

Seam Properties Dialog Box

Specifies the properties for the seam that you are placing or editing.

Main Tab (Seam Properties Dialog Box) (on page 146)

Relationship Tab (on page 34)

Configuration Tab (on page 32)

General Tab (on page 33)

Main Tab (Seam Properties Dialog Box)

Specifies the general properties of the seam.

Name

Specifies the name of the seam.

Rule

Select the naming rule to use to name the seam.

- Select **StdHierarchyChildNamingRule** to use the following syntax: <Parent Name>-<"IJSeam"><Index Number>. For example, D0-1DCK-1-IJSeam1, where D0 is the reference plane, 1DCK is an index number appended by the plate type (Deck in this example), 1 is the Workshare Location ID, and IJSeam1 is "IJSeam" appended by an index number.
- Select **User Defined** to name the seam using the appropriate box.

Type

Specifies the type of seam that you are placing: **Design Seam** or **Planning Seam**. The list is defined by the StructSeamType codelist.

Landing Curve Definition Method

Displays how the landing curve for the seam was defined.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

SECTION 18

Run Report

You can generate reports (that represent the information available on the **Common Part Explorer** as a workbook) of various common parts linked to a various **Planning Assembly**.

1. Click **Tools** > **Run Report** > **Catalog Reports** > **Planning** > **Common Parts Reports.**
2. Save the file.
3. Click **Run.**

SECTION 19

Assembly Views

Show Selected

The **Tools** > **Show Selected** command allows you to quickly hide all but the selected items, turn on the display of selected items, and scale the view of the selected items to fit in the active window. The command is only active when one or more items are selected.

The command is only active when a single assembly is selected. When the show selected operation completes, the software automatically returns to the previous active command.

When designing steel, outfitting, or assemblies, you are often in a situation where the graphic display is blurred or obstructed by objects with no interest to the task being performed. The **Show Selected** command lets you quickly focus on objects of interest to a specific task. By selecting objects graphically, or in the **Workspace Explorer,** and then activating the command, the display of selected objects is turned on, the display of all other objects is turned off, and selected objects are scaled to fit within the active window.

Show selected objects

1. Select the items that you want to show.

   TIP You can select the item in the model or in the **Workspace Explorer**.

2. Click **Tools > Show Selected**.

   TIP If you press **Shift** while you execute the command, the resulting view does not fit to the active window.

Show Parts

The **Tools** > **Show Parts** command allows you to quickly view the parts that are directly assigned to the selected assembly or block. Typically, these parts are considered as candidate parts for a new subassembly. The display of all parts of the selected assembly is turned on and the display of all other items is turned off. For blocks, the parts within the **Unprocessed Parts** folder are also displayed.

The command is only active when a single assembly is selected. When the show parts operation completes, the software automatically returns to the previous active command.

In the Planning task, you work with decomposing the model structure into blocks and assemblies. The task is, to a certain level of decomposition, done from the top down by gradually splitting the model into smaller and smaller blocks. At some point, you start composing assemblies from the bottom up. With the **Show Parts** command, you have the ability to see all the atomic parts assigned to the assembly without seeing the parts that are in the sub-assemblies. You can consider these parts as candidates for a new subassembly.

When evaluating a proposed building method, the **Show Parts** command can also be used to quickly see the graphical representation of parts added to an assembly at a specific level of the assembly hierarchy.

Show parts

1. Select the assembly for which you want to show the components parts.

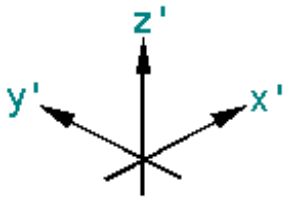
   TIP You can select the object in the model or in the **Workspace Explorer**.

2. Click **Tools > Show Parts**.

   TIP If you press **Shift** while you execute the command, the resulting view does not fit in the active window.

Show Assembly View

The **Tools** > **Show Assembly View** command allows you to quickly display an isometric view of the selected block, assembly, or assembly block based on the local coordinate system created when its manufacturing orientation was assigned. The coordinate system has the manufacturing up-vector z'-axis and the x'- and y'-axis from the assigned viewing rotation. In the following illustration, an isometric projection of a coordinate system shows the main directions of the coordinate axes.



The resulting display of the selected assembly or block is scaled to fit in the view of the active window and hiding all items not pertinent to the selected assembly. If clipping exists, it is cleared.

The **Show Assembly View** command is only active when you have a single block, assembly, or assembly block selected.

In the Planning task, you work with decomposing the model structure into assemblies designed for a specific workcenter. The task of designing assemblies and building methods includes deciding the manufacturing orientation of assemblies (the orientation, or up-vector, of an assembly while it's being built). The **Show Assembly View** command helps you to evaluate assemblies and building methods.

Show the assembly view

1. Select the assembly or block for which you want to display the assembly view.

   TIP You can select the item in the model or in the **Workspace Explorer**.

2. Click **Tools** > **Show Assembly View**.

TIPS

- The command is only active with a single assembly or block selected.
- If you press **Shift** while you execute the command, the resulting view is not fit to the active window.

NOTES

- When you execute the **Tools > Show Assembly View** command, the software displays an isometric view (after the ISO 5456 part 3 standard) of the selected assembly/block, scaled to fit the active view. All other items are hidden.
- When the command terminates, you return to the previous active command.

Show Assembly Sequence

The **Tools** > **Assembly Views > Show Assembly Sequence** command uses assembly sequence information and manufacturing orientation (the orientation, or up-vector, of an assembly while being built) to show a step-by-step animation of the assembly sequence for a selected block and/or assembly. The animation displays in a separate window with a new private graphic view.

When the command is executed, the software displays a new window that includes the active view of the selected assembly or block from the **Main** window. During the animation, the software scales the view to fit the selected assembly. As the software moves through each step in the animation, items are added according to the defined assembly sequence. Grouped objects are added together; objects with no sequence information are added last. You can automatically run the sequence as defined using the **Sequence**  command. For more information, see *Sequence* (on page 79). You can also manually define a sequence using the ribbon.

Show Assembly Sequence Ribbon

Stage

Shows the sequence number of the displayed assembly stage. You can select from a list of actual sequence numbers for the selected assembly. The items added at the listed stage display as graphically distinct from assembled items and are distinguishable in the **Workspace Explorer**. Use this option to jump from one sequence stage to another.

Play Sequence

Indicates how the software displays the sequence. You can switch between **Manual** and **Auto** at any point in the sequence.

- **Manual** indicates that the software displays a sequence you specify using the controls on this ribbon. This is the default action.

- **Auto** indicates that the software displays the sequence specified using the **Sequence** command or the **Sequence Form** . For more information, see *Sequence* (on page 79). You can also specify the **Mode** and **Interval**.

Mode

Indicates how the automatic play sequence plays nested subassemblies. This option is only available when **Play Sequence** is set to **Auto**.

- **Active** indicates that the software plays only the active assemblies.
- **Nested** indicates that the software steps into all nested subassemblies.

Interval

Indicates the length of time that the software displays each part. This option is only available when **Play Sequence** is set to **Auto**.

Play

Plays the sequence. This option is only available when **Play Sequence** is set to **Auto**.

Pause

Pauses the sequence. If you click **Play** after you click **Pause**, the software resumes the sequence where you paused. This option is only available when **Play Sequence** is set to **Auto**.

Stop

Ends the sequence. If you click **Play** after you click **Stop**, the software starts from the beginning of the sequence. This option is only available when **Play Sequence** is set to **Auto**.

Back

Moves the assembly sequence to the previous sequence number. All items not represented at the previous stage are removed from the private graphic view. This option is only available when **Play Sequence** is set to **Manual**.

Forward

Moves the assembly sequence to the next sequence number and shows all items represented at the next sequence stage. This option is only available when **Play Sequence** is set to **Manual**.

To Start

Moves the assembly sequence to the initial sequence number. All items not represented at the initial stage are removed from the private graphic view. This option is only available when **Play Sequence** is set to **Manual**.

To End

Moves the assembly sequence to the final sequence number and shows all items represented at the final stage. The view that displays is identical to the one produced with the **Tools > Show Assembly View** command. This option is only available when **Play Sequence** is set to **Manual**.

Step Out

Moves the assembly sequence out of a nested subassembly. This option is only available

when **Play Sequence** is set to **Manual**.

Step In

Moves the assembly sequence into a nested subassembly. This option is only available when **Play Sequence** is set to **Manual**.

Previous Stage

Moves the assembly sequence to the previous stage in the nested subassembly. This option is only available when **Play Sequence** is set to **Manual**.

Next Stage

Moves the assembly sequence to the next stage in the nested subassembly. This option is only available when **Play Sequence** is set to **Manual**.

Type

Controls how the software displays weight and dimension properties.

- **Accumulated** indicates that the software lists accumulated weight and dimensions of the items contained by the assembly at the current stage.
- **Sequence** indicates that the software lists the properties of the items being added at the current stage.
- **Assembly** indicates that the software lists only the properties for the selected assembly.

Weight

Lists the weight of all subparts and subassemblies according to the **Type** option.

Length

Lists the combined range box length of all subparts and subassemblies according to the **Type** option.

Width

Lists the combined range box width of all subparts and subassemblies according to the **Type** option.

Height

Lists the combined range box height of all subparts and subassemblies according to the **Type** option.

View Direction

Specifies the view orientation of the selected assembly.

Weight & Range

Displays or hides the **Weight**, range (**Length**, **Width**, and **Height**), and **Type** property information. The software does not calculate weight or range if **Weight & Range** is not selected.

Properties

Displays or hides the assembly sequence property information.

Child

Displays the part name of the currently playing part.

Assembly

Displays the properties associated with the currently selected assembly.

View the assembly sequence

1. Select the assembly or block whose assembly sequence you want to view.

   TIPS

   - You can select the block from the graphic view or the **Workspace Explorer**.
   - You can select an assembly or assembly block only from the **Workspace Explorer**.

2. Click **Tools > Assembly Views > Show Assembly Sequence.**
3. On the **Show Assembly Sequence** ribbon, use the **Stage** list to view the various stages of the assembly sequence.

   TIP Alternatively, you can use the **Forward**, **Back**, **To Start**, and **To End** controls to move through the assembly sequence.

4. You can use the **Type** list to control what properties are displayed at the selected stage in the assembly sequence.

NOTES

- When you execute the **Show Assembly Method** command, the software displays an active view of the **Main** graphic window for the selected assembly in its manufacturing orientation, scaled to fit the current view in a separate window. All other items are hidden.
- As you run through the animation sequence step by step, items are added to the view in the active window. Grouped objects are added together.
- Items with no sequence information are added last, all at one time.

SECTION 20

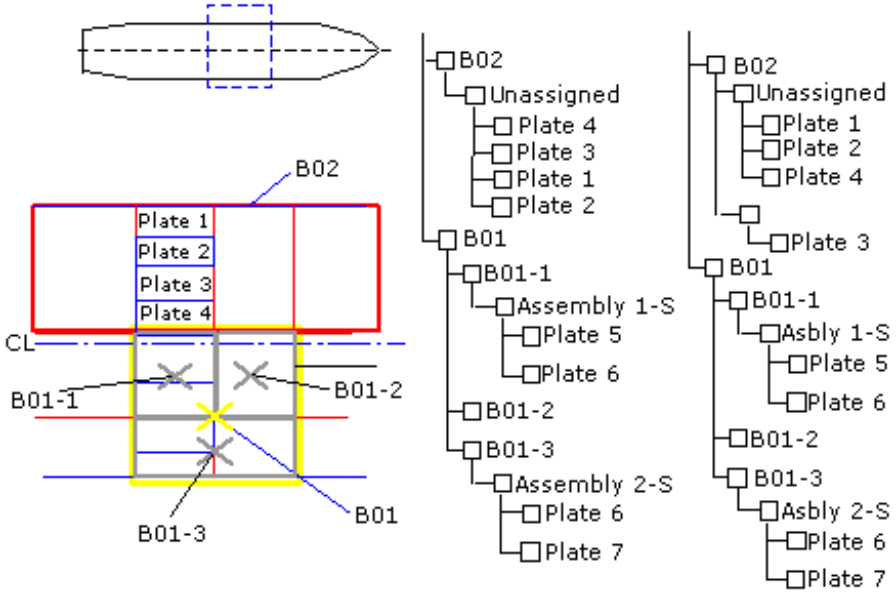
Assembly Arrangements

Synchronize Hierarchy

The **Tools** > **Synchronize Hierarchy** command allows you to perform a bidirectional synchronization of unassigned parts in symmetrically related blocks and assemblies. Using a select set of symmetrically related assemblies or leaf level blocks, the command locates parts that have a symmetrical counterpart still awaiting manual assignment. When located, these unassigned parts are moved from the **Unassigned Parts** folder to the symmetrically related assembly.

IMPORTANT

- Parts that have a symmetrical counterpart with an assignment different than the **Unprocessed Parts** folder are not reassigned by this synchronization mechanism. No master-slave behavior exists.
- The **Tools** > **Synchronize Hierarchy** command does not synchronize block structure. You must manually ensure that leaf block definitions are symmetrical. If the two assemblies/blocks are not symmetrically defined, the software searches for unassigned symmetric parts and creates assemblies if there are symmetric parts to assign. In the following illustration, note that the block structure in B01 is not copied into B02 when synchronization process is attempted:



Synchronize Assembly Hierarchy Selection Dialog Box

Automatically include lower level symmetrical assemblies and blocks

When enabled, all lower level assemblies and blocks are automatically checked by the

software.

Synchronize user-defined assembly properties

When enabled, the software attempts to synchronize all user-defined properties. For each assembly that is out of synchronization, the software prompts you to decide which assembly to use as the source for property information. The default source is the latest assembly modified.

Select hierarchy and grid

Displays an assembly hierarchy structure that allows you to select assemblies and blocks. Only assemblies and blocks are displayed in the **Name** column, and only assemblies and blocks with a symmetry relationship formed to another assembly/block are selectable.

Select all

Selects all checkboxes in the **Select hierarchy and grid** box. You can use the cursor to manually uncheck assemblies and blocks.

Clear all

Clears all checkboxes in the **Select hierarchy and grid** box.

Symmetrical assembly

Displays the name of the symmetrical assembly when an assembly is selected. If there is no symmetrical assembly, "No mirror assembly" appears.

OK

Closes the dialog box and processes the synchronization. Also, a refresh of the workspace is done in order to reflect changes in the defined workspace.

Cancel

Cancels the select set and closes the dialog box without processing the synchronization.

Apply

Applies the synchronization to the select set without closing the dialog box.

Synchronize the assembly hierarchy

1. Click **Tools > Synchronize Hierarchy**.
2. In the **Synchronize Assembly Hierarchy Selection** dialog box, use the boxes in the **Select hierarchy and grid** box to select each assembly or block to be synchronized.

   TIPS

   - When you select an assembly or block node in the grid, it is highlighted in the graphical window.
   - As you mouse-over an assembly or block, the symmetry value is displayed.
3. Click **OK** to process the synchronization and close the dialog box.

TIP You can click **Apply** to process the synchronization and keep the dialog box open.

NOTE Right-clicking on an assembly/block node in the **Select hierarchy and grid** box displays a shortcut menu with two options regarding the formation of symmetry relationships - **Form relationship** and **Remove relationship**. If the selected node is already symmetrically

related, the relationship can be removed. If the selected node is not related, both options are grayed out; however, if two previously non-related nodes are selected before you perform the right-click, a relationship can be formed.

Block Assignment

The block assignment process assigns and reassigns parts to blocks. When the process is activated, the software automatically and continuously queries the database for new and modified parts to assign and reassign to a block.

You can also specify which types of parts should be processed and how they should be processed.

For the block assignment process to run automatically and continuously, a dedicated server or workstation computer is required. If a dedicated computer is available, you can initiate the block assignment process manually at regular intervals.

Initial Block Assignment of Parts

The purpose of Block Assignment is to take the first cut at where parts fall in the construction planning process. When the block assignment process is running, the software assigns a new part to a block based on the spatial location of the part and the spatial definition of the block. Parts are assigned to blocks at the deepest level possible in the assembly hierarchy within which the part is fully contained. If the part intersects with lower-level blocks and is within an overlap factor assigned in the rules, the part is assigned to the appropriate block. For example, if the overlap factor is 0.8, and at least 80% of a part is within a block, the part is assigned to that block. Otherwise, the software places the part on the list of intersections for the involved blocks and block cutting planes. Only parts that are in working status are considered by Block Assignment.

Each level of the hierarchy contains its own **UnAssigned Parts** folder. When parts are assigned to blocks, those parts are placed in the **UnAssigned Parts** folder for that level until they are assigned. In a completely planned ship, most parts will be ultimately assigned to assemblies that are children of the blocks.

After you manually assign a part to an assembly within or directly under the block, outside of the **UnAssigned Parts** folder, the part falls out of the domain of the assignment process. However, as long as the parts remain in the **UnAssigned Parts** folder, the block assignment process will reprocess them whenever a part or block is modified.

Modification of Parts

Whenever you modify a part in the **UnAssigned Parts** folder and an intersection with a block boundary occurs, the software places the part on the list of intersections and sets its status as **Assigned** to the block from which it originated.

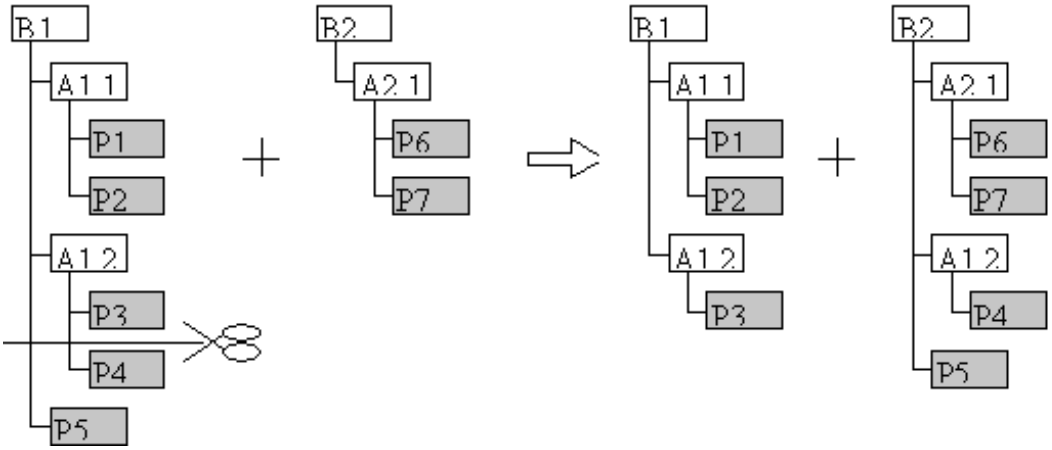
When you modify a part that exists outside of the **UnAssigned Parts** folder, the assembly hierarchy is left unchanged. If an intersection with a block boundary occurs, the software places the part on the list of intersections and displays its status as **Assigned**. If you modify a part and as a result it is fully contained within a new parent block, the software considers the part as new and leaves it in the **UnAssigned Parts** folder.

Creation and Modification of Blocks

When you create or modify blocks, the block assignment process checks the parts in the **UnAssigned Parts** folder to determine whether a part can be moved to another block or listed as an intersecting part.

When parts have been assigned to assemblies, defining or modifying a block boundary is more complicated. Consequently, Intergraph PPM recommends that as much of the designed building method be preserved as possible. Defining or moving block boundaries removes disassociated parts, assemblies, and part of assemblies from the block in which they were originally assigned during the initial design of the assembly hierarchy.

To preserve the design decisions documented in the assembly hierarchy, the block assignment process only moves parts to the block in which fully contained and duplicate parts of the assembly hierarchy are represented in the original block. The duplication includes the assigned assembly properties. If the duplication involves a whole branch of the assembly hierarchy, that branch is deleted from the original block. In the following illustration, Block B1 is reduced in size by the modification of a block cutting plane. The neighboring block, Block B2, is increased in size by the operation. Parts are moved and assembly A1.2 is duplicated to block B2:



If a part intersects with the block boundary between previously well-known blocks (block cutting plane modification), the part is left in the block from where it originated, and the status in the list of intersections is set as **Assigned** in that block. If the part intersects with one or two blocks that were previously unrelated, the software places the parts in the **UnAssigned Parts** folder of the parent block and lists them among the intersecting parts for the involved blocks or block cutting planes. Assemblies that are emptied of parts are deleted.

NOTE You can set a percentage tolerance to assign a part to a block that contains the greater portion of the part. When part that is inside a block has a percentage volume greater than the specified tolerance, it is assigned to this block. For information on how to set tolerance, see *Planning Reference Data Guide*.

Scheduling Block Assignment with Batch Services

Planning allows you to schedule some tasks during nonproductive hours, for example, nights or weekends, using Intergraph Batch Services.

NOTE For more information about Intergraph Batch Services, see the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

Block Assignment Dialog Box

Allows you to start and stop the block assignment process and specify on which server the process is to run.

What do you want to do?

- *Start the block assignment process* (on page 158)
- *Stop the block assignment process* (on page 159)
- *Schedule block assignment using Batch Services* (on page 159)
- *Create block assignment log file* (on page 160)

Start the block assignment process

1. Click **Tools > Assembly Arrangements > Block Assignment**.
2. On the **General** tab of the **Block Assignment** dialog box, click **Browse** and locate the server on which the block assignment process is to run.
3. Select the part location: **Database** or **Workspace**.
4. In the **Part selection** box, select how you want the parts processed: **By Time**, **By Block** or **By Filter**.
   - If you select **By Time**, set the **Query range** options:
     i. Select **Resume automatic query** to resume the previous block assignment process that was halted.
     ii. Select **User-defined query range** and type the start date and time and end date and time in the appropriate fields.
     iii. Define the **Number of grace minutes**.
   - If you select **By Block**, click in the **Block** box and navigate to the appropriate block.
   - If you select **By Filter**, click in the **Filter** box and navigate to the appropriate filter.
5. Click **Start** to activate the block assignment process.
6. Check **Run in background** to close the dialog box.

NOTES

- If the software detects that the server you selected in step 2 is not running the appropriate software, you are prompted to select another server.
- You must have read and write permissions to the model database to start and stop the block assignment process. For more information about access permissions, see your Smart 3D administrator.

Stop the block assignment process

1. Click **Tools > Assembly Arrangements > Block Assignment**.
2. Click **Stop search process**  to stop the block assignment process.
3. Click **Close** to close the dialog box.

NOTE You must have read and write permissions to the model database to start and stop the block assignment process. For more information about access permissions, see your Smart 3D administrator.

Schedule block assignment using Batch Services

IMPORTANT When setting up block assignment to run in batch mode, the following exceptions apply for defining options on the **Block Assignment** dialog box:

- The batch process considers only objects from the database and ignores the **Workspace** option if it is selected.
- Unlike the interactive block assignment process that you can run indefinitely using the **Resume automatic query** option, a batch job can be run only for a definite period of time and will ignore the **Resume automatic query** option if is it is selected.
- A batch job takes **Number of grace minutes** as **1** regardless of the value that you type.
- The **Run In Background** option is not relevant to a batch job and is ignored if it is selected.

1. Click **Tools > Block Assignment**.
2. On the **General** tab of the **Block Assignment** dialog box, use the **Part selection** box to specify how you want the parts processed: **By Time**, **By Block**, or **By Filter**.
   - If you select **By Time**, set the **Query range** option as **User-defined query range** and type the start date and time and the end date and time in the appropriate fields.
   - If you select **By Block**, click in the **Block** box and navigate to the appropriate block.
   - If you select **By Filter**, click in the **Filter** box and navigate to the appropriate filter.
3. Click **Submit Job**.
4. Complete the **Schedule Block Assignment** dialog box to configure batch processing, and then click **OK**.

   The software displays the message, "This job has been scheduled to run on the chosen server."
5. Click **OK** to dismiss the message, and then click **Close**.

NOTES

- For more information about SmartPlant Batch Services, see the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.
- To start and stop the block assignment process, you must have read and write permissions to the Model database. For more information about access permissions, see your Smart 3D administrator.

Create block assignment log file

1. Open the **Planning Registry Settings** dialog box: [Product Folder]\Planning\Tools\Developer\Bin\PlnRegSettings.exe.
2. Check **Log to File**.
3. Set the **Output Filename**.

NOTE When you run the block assignment process in batch mode, the software automatically creates the log file BlockAssignLog.log and saves it in the Temp folder. When you run the block assignment process interactively, this feature is not set by default; you must set it.

Schedule [Task] Dialog Box

Queue

Displays the name of the queues configured by an administrator for the job. For more information on configuring the queues, see *Configure Queues for Jobs* in the *Project Management User's Guide*.

Run job

Sets the frequency with which the job runs. Jobs can be scheduled to run once or on a regular interval (daily, weekly, or monthly). Depending on the job frequency selected, additional controls display. These controls allow you to define more specific scheduling information. The scheduling controls can be changed only at job submission.

Run on

Sets the time to start running the job.

Options

Opens the *Optional Schedule Properties Dialog Box* (on page 161) that you can use to define a start and end date.

Run on box

Contains a calendar from which you can select the run date. This option is available when you select **Once** from **Run job**.

Every X days

Specifies how many days pass between job runs. This option is available when you select **Daily** from **Run job**.

Every X weeks

Specifies how many weeks pass between job runs. In addition, you can select on which days the job runs. This option is available when you select **Weekly** from **Run job**.

Day X of the month

Specifies on which day of the month the job runs. This option is available when you select **Monthly** from **Run job**.

The X Y of the month

Specifies on which day of the month the job runs. For example, you can select the last Monday of the month. This option is available when you select **Monthly** from **Run job**.

Job Start

Notifies you when the job starts, if Outlook is set up.

Job Completion

Notifies when the job completes, if Outlook is set up.

Job Abort

Notifies you if the job aborts, if Outlook is set up.

Address Book

Selects the name of the person to be notified by e-mail of the job status, if Outlook is set up. If Outlook is not available, this option does not work. You can also type the address manually. The person you define here receives an email with the job log files after the job finishes.

NOTES

- The Batch Services SMTP option must be configured on the batch server for this to work. For more information, see the Intergraph Smart Batch Services documentation.
- The WinZip application is no longer required on the batch server to compress any emailed attachments. Compression is now done with functionality included in Smart 3D.

Optional Schedule Properties Dialog Box

Provides more options on the **Schedule Backup** dialog box. This dialog box opens when you click **Options**.

Start date

Sets an optional start date.

End date

Sets an optional end date, if checked.

Import Assembly Data from Excel

Imports an assembly hierarchy defined in a Microsoft Excel spreadsheet. You can also delete or assign parts to assembly hierarchies. Smart 3D delivers an example spreadsheet to the *SharedContent*\Import\Planning\S3D_ImportAssemblyData.xlsx file. The software reads the **Process** sheet from this spreadsheet to determine the actions to take. This command is available on the **Tools > Assembly Arrangements** menu.

Import Assembly Data from Excel Dialog Box

Controls parameters for importing, deleting, or assigning parts to an assembly hierarchy based on input from an Excel spreadsheet.

Excel files

Displays a list of Excel spreadsheets for the software to process. Smart 3D delivers an example spreadsheet to the *[Reference Data Folder]*\SharedContent\Import\Planning\S3D_ImportAssemblyData.xlsx file.

Add

Displays the **Open** dialog box so that you can browse to an Excel spreadsheet and add it to the list.

Delete

Removes the highlighted spreadsheets from the list.

Open

Opens the highlighted spreadsheet in Microsoft Excel.

Log file

Displays the path and file name of the log file for the import process. You can select a recently used file from the list, or click **...** to browse to a file.

Import option

Displays the available import actions as defined in the spreadsheet. Select the actions for the software to perform. Clear the check box to skip those actions.

Import

Performs the options as specified in the **Import option** list.

Review Log

Displays the log file as defined in the **Log file** box.

Reset

Returns the **Import option** list to its default condition.

Cancel

Dismisses the dialog box.

SECTION 21

Common Parts

A common part is a part that is determined to be identical based on user-defined criteria to one or more other parts. From manufacturing perspective, any one of these identical parts can be used and no unique part identification needs to be made or managed.

Common Part Service

The **Common Parts Service** identifies and manages structural parts that are common within the scope of a block, an assembly, or an assembly block**. Common Part Definition** sets rules for comparison. The process creates groups and folders. The Common Part Service dialog can be found under the Tools menu under **Tools > Common Parts > Common Part Service**.

Common Part Definition

Common Part Definition

Displays the common part definitions to be selected.

View Setting

Activates the **View Settings** dialog box.

Manage Filters

Show Existing

Displays the managers associated to the **Common Part Definition** selected in grid**.**

Database

Displays the associated managers present in the database.

Workspace

Displays the associated managers present in the workspace.

Add Rows

Adds new rows to the Common Part Service.

Process Status

Object Type and Sub-manager being processed

Displays the object type and sub-manager being processed for the assembly that is being processed.

Number of sub-assemblies processed

Displays the number of sub-assemblies processed in the assembly that was processed.

Estimated time remaining

Displays the estimated time to complete the sub-assembly processing.

Process Log

Enable Log

Select to generate a detailed log file.

Browse

Allows you to browse and to save the log file.

Submit Job

Allows you to submit the selected job.

Start

Starts the selected job.

Stop

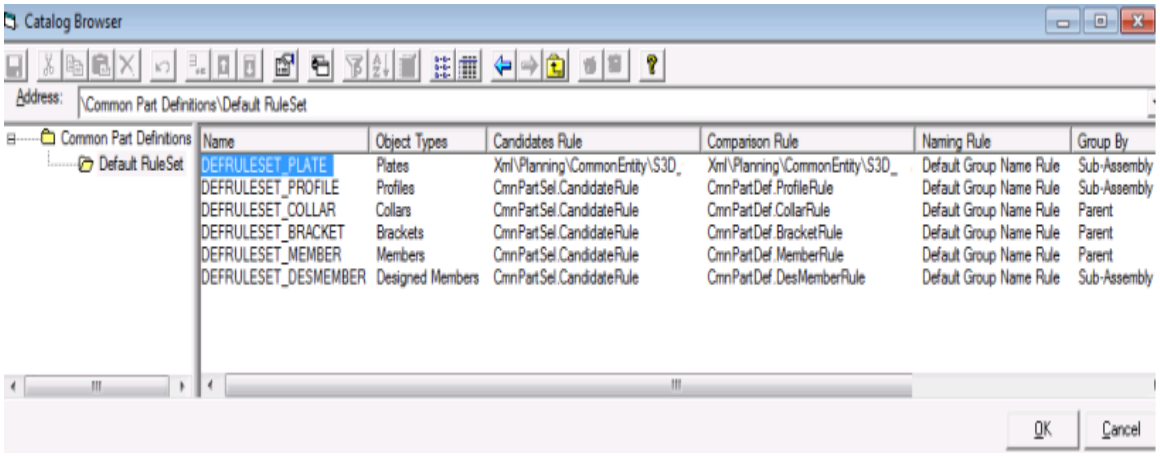
Stops the selected job.

Close

Closes the log file.

Common Part Definition

A **Common Part Definition** is a set of rules that is divided into **Candidates**, **Naming** and **Comparison Rule**.



Name

Name of the rule.

Object Types

Type of object to which the rule applies.

Candidates Rule

Determines the set of parts that has to be processed to find common parts. The rule can be an XML file that specifies how to identify the candidates. The rule serves the following purposes:

- Excludes any child parts that you do not want to evaluate.

- Returns the set of parts or assemblies the service should evaluate.

Comparison Rule

Consists of logic for comparison. The rule can be an XML file that specifies the properties to be compared, and how to compare them.

Naming Rule

Names the part.

Group By

Specify how to group the parts. Available choices are:

- **Sub-Assembly** - Each sub-assembly under the selected assembly will have its own group(s) of common parts of a particular object types. The immediate child parts of each sub-assembly are compared among themselves to find common parts.
- **Parent** - All child parts at various levels under the selected assembly are evaluated and groups created are associated to the assembly.

NOTE The figure above only shows plates, profiles, collars, brackets and members, you can add rules to an existing **Common Part Definition** using bulkload service. You can also add new **Common Part Definitions**.

Start Common Part Service

1. Click **Tools** > **Common Parts** > **Common Part Service.**

   *The **Common Part Service** dialog box is displayed with the first available **Common Part Definition** in the selected catalog.*

2. Click **Common Part Definition** > **More**, and select the required **Common Part Definition** from the **Catalog Browser**.
3. Click **OK** to close the **Catalog Browser**.
4. Select the required option under **Manager Filters** > **Show Existing Managers** or **Database**, or **Workspace**.

   *The common part managers associated to the selected **Common Part Definition** are displayed.*

5. Click **Add Rows** to add rows, and select the new assemblies you want to process.
6. Check the required object types for assemblies you want to process.
7. Check or clear the **Discard existing Common Parts** option. When checked, this will delete the common part managers in the selected assembly and create a new one.

   NOTE You can click **View Settings** to view the rule settings for all the object types that have the XMLs specified.

8. Submit a job or start the service.
   - **Submit Job** will let you schedule a batch job. You can submit a job on the local machine (default) or a remote machine.

- **Start** will start the process immediately. You can see the status on the dialog box.

NOTES

- On closing the dialog box, the process either runs in the background or starts at a scheduled time. When you open the dialog box again, the process status is shown (if the process is still running). **Process Status** estimates the remaining time during service.
- You can generate a process log by saving the log file to a specified path by using **Process Log** > **Browse,** and clicking **Start**.
- When you start the **Common part Service** for strake seam split parts, make sure you set the definition check type to skipped.

  Example:

  <S3D_Check NAME="Definition" KEYWORD="Definition" TYPE="Skipped" CATEGORY="Property" ALLOWABLETYPES="Attribute,Rule-based,Skipped"/>

  This is required to include parts split by strake seam as candidates for the common part service.

Common Part Explorer

Common Part Explorer displays the common parts. The parts are displayed as groups under folders of the relevant block, assembly or assembly block. The groups are further divided according to object types.

After the **Common Part Service** is complete, the following groups and folder are created.

Common Part Manager

Object managers called the **Common Part Managers** are displayed in the **Workspace Explorer** to notify that the objects were processed.

Common Part Folder

Each group consists of **Common Parts Folders** that are named as per the object types, and a separate folder called the **Uncommon Parts** for parts that are not common.

Common Part Group

Each part that is identified as common is given a common part name based on a naming rule. The service does not change the existing name of the part. The parts are displayed as groups under folders of the relevant block, assembly, or an assembly block. All groups are further grouped according to the type of objects called the **Object Types**.

Excluded Common Parts

Common parts that are removed from a common part group are moved to this folder.

Relocated Common Parts

This folder contains parts that can be relocated. The parts are moved to this folder under the following conditions:

- When the parent assembly of a part is assigned to a new assembly and the common part definitions associated to the assemblies are different.
- When a part is moved to an assembly that has common parts.
- When a part is removed from the **Excluded Common Parts** folder.

What do you want to do?

- *View Common Parts* (on page 167)
- *Remove Common Parts* (on page 167)
- *Modify Common Part Groups* (on page 167)

View Common Parts

1. Select the block, assembly or assembly block to view the common parts.
2. Click **Tools** > **Common Part Explorer** or right-click on the block, assembly or assembly block in the **Workspace Explorer,** and click **Common Part Explorer.**
3. Click **Workspace** or **Database** to view all the common parts**.** To highlight all the parts under a group, select **Highlight Group** and hover the mouse on the common part group.

   NOTE If you want the parts to fit in graphic view, right-click the common part group, and select **Fit**.
4. Click  , or right-click in the white space of the **Common Part Explorer**, and then click **Refresh**.

   *All objects in the **Common Part Explorer** are updated.*

NOTES

- Parts that are not common are grouped under the folder **Uncommon Parts**.
- Groups are classified according to object types. Comparison rules check for all properties such as inner and outer contours, material grade, and thickness.

Remove Common Parts

1. Right-click a common part.
2. Click **Remove From Group.**

   NOTE The part is moved to **Excluded Common Parts** folder.

Modify Common Part Groups

1. Right-click a common part group.
2. Click **Approve Group** or **Remove Group**.

NOTES

-  - Indicates that the common part group is out of date.
- **Approve Group** - Freezes the common part group.
- **Remove Group** - Dissolves the common part group and moves the parts to the **Excluded Common Parts** folder.
- **Update Group** - Updates the common part group by re-processing the selected group.

- **Fit** - Fits the common parts that are under the selected common part group in the active view.

Reevaluate parts for common part managers

1. Right-click a common part manager.
2. Click **Reevaluate Parts**.

   *The **Common Part Service** dialog box is displayed.*
3. Use **Common Part Service** to reevaluate the parts. For more information, see *Start Common Part Service* (on page 165).

Update a common part manager

1. Right-click the common part manager.
2. Click **Update Manager**.

   The common part manager is re-processed and updated.

Update a common part folder

1. Right-click the common part folder.
2. Click **Update Folder**.

   The common part folder is re-processed and updated.

Common Part Manager Properties Dialog Box

Specifies the properties for object managers.

See Also

General Tab (on page 168)
Relationship Tab (on page 34)
Configuration Tab (on page 32)
Notes Tab (on page 33)

General Tab

Displays and defines the general property names and values of the selected **Common Part Manager**.

Standard Category

Name

Displays the name.

No. of Common Part Groups

Displays the number of common part groups.

Status

Displays if the status is running or not.

Common Part Definition

Displays the definition.

Common Part Folder Properties Dialog Box

Specifies the properties for object folders.

See Also

General Tab (on page 169)
Relationship Tab (on page 34)
Configuration Tab (on page 32)
Notes Tab (on page 33)

General Tab

Displays and defines the general property names and values of the selected **Common Part Folder**.

Standard Category

Name

Displays the name of the folder.

Unprocessed Parts

Displays the number of parts that are out of date, parts that have been modified after the service is processed.

Excluded parts

Displays the number of excluded parts.

Common Part Utilities

Common Part Utilities are used to view and edit comparison checks for object types or XML files. Common Part Utilities can be found under the **Tools** menu.

Test Rules

The **Test Rules** command compares the values mentioned in the user-defined properties of an object type or an XML file and displays the result.

Test Rules Ribbon

Mode

Specifies what is being compared. Select an object type or an XML file.

Comparison Rule

Displays the path of the object type or the XML file.



Select Parts to Compare

Allows you to select the parts to compare.

Accept

Accepts the selection.

Reject

Rejects the selection.

Results

Opens the **Rule Results** dialog box. This option is enabled when the **Mode** is selected.

What do you want to do?

- *Compare an object type* (on page 170)
- *Compare an XML* (on page 171)

Compare an object type

1. Click **Tools** > **Common Part Utilities** > **Test Rules**.

   *The **Test Rules** ribbon is displayed.*

2. Click **Object Type** in the **Mode** list.

   *The **Common Part Definition** dialog box is displayed.*

3. Select the required rule from the second column of the dialog box.
4. Click **OK**.

   *The **Comparison Rule** is updated, and the **Select Parts to Compare** command is enabled.*

5. Selects the parts to compare.
6. Click **Accept**.
7. Click **Results**.

   *The **Rule Results** dialog box is displayed.*

8. Click **ProgID** to view the result.

NOTES

- The result also displays the status **All parts are common** or **Found some un-common parts**.
- The properties of the un-common parts are highlighted in red.
- You can use the **Save Results** option to save the results.
- The **Results** command is not displayed if the rule does not use an XML file. The **Compare** command is displayed in this case. When you use **Compare**, the selected objects are compared and a message is displayed if the parts are common or not.

Compare an XML

1. Click **Tools** > **Common Part Utilities** > **Test Rules**.

   *The **Test Rules** ribbon displays.*

2. Select **XML** under the **Mode** drop-down list.

   *The **Select an XML file** window is displayed.*

3. Select the required XML.
4. Click **Open**.

   *The **Comparison Rule** updates, and the **Select Parts to Compare** command is enabled.*

5. Select the parts to compare.
6. Click **Accept**.
7. Click **Results**.

   *The **Rule Results** dialog box displays.*

8. Select the **ProgID** to view the result.

NOTES

- The result also displays the status **All parts are common** or **Found some un-common parts**.
- The properties of the un-common parts are highlighted in red.
- You can use the **Save Results** option to save the results.

Edit Rule Settings Dialog Box

Select ruleset

Specifies the **Common Part Definition**.

Select object type

Allows you to select an object type. You can select **Plates**, **Profiles**, **Collars**, **Brackets**, or **Members**.

Rule Path

Displays the path of the XML file of the selected rule set.

See Also

Candidate Tab (Edit Rule Settings Dialog Box) (on page 172)
Comparison Tab (Edit Rule Settings Dialog Box) (on page 172)
Rule Variables (Edit Rule Settings Dialog Box) (on page 195)

Candidate Tab (Edit Rule Settings Dialog Box)

Node

Displays the specific purpose served by the candidates rule.

- **Assembly** indicates that the corresponding row is used to specify how sub-assemblies are identified.
- **Candidate** indicates that the row is used to specify how candidates are identified.

Type

Displays the filter or the rules used.

Value

Displays the **ProgID** of the rule or the path of the filter used. You can change the **ProgID** by editing the text, and using the **More..** option to change the filter.

Comparison Tab (Edit Rule Settings Dialog Box)

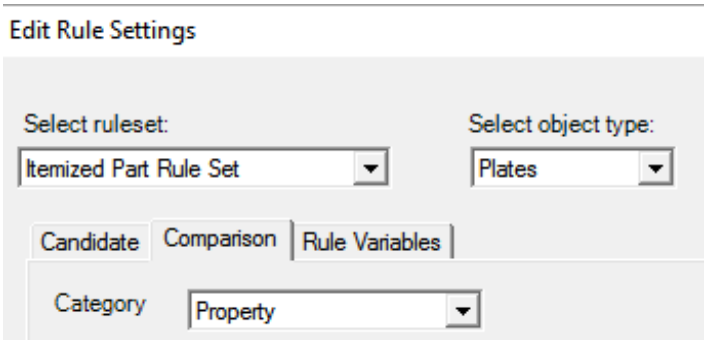
Category

Displays the category of the comparison check. You can select **Property**, **Geometry**, **Connection**, **Edges**, or **Manufacturing**.

Check Display Text

Displays the name and the type of the comparison check. See descriptions for each **Check Display Text** category below based on each object and category combination.

Plates Type / Property Category Combination



Check Display Text

- **Plate Material** - Compares the material type and grade like steel, wood, and so forth of the candidate and target plates.
- **Plate Thickness** - Compares the thickness of candidate and target plates.
- **Plate Definition** - Compares candidate and target part definitions. This option is not applicable for plates.
- **Naming Category** - Compares the naming category such as longitudinal strength, deck, and so forth, of candidate and target plates.

- **Plate Type** - Compares the plate type such as deck, bulkhead, and so forth, of the candidate and target plates.
- **Plate Tightness** - Compares candidate and target plate tightness.
- **Plate Curvature Type** - Compares the plate curvature type, such as flat, curve, and so forth, of candidate and target plates.
- **Plate Area** - Compares the area of candidate and target plates.
- **Plate Length** - Compares candidate and target plate lengths.
- **Plate Width** - Compares candidate and target plate widths.
- **Dry Weight** - Compares the dry weight of candidate and target plates.
- **Wet Weight** - Compares the wet weight of candidate and target plates.
- **Building Method** - Compares the build method, such as slide, vertical drop, and so forth, of candidate and target plates.
- **Slot Connectivity** - Compares the slot connectivity, such as web left, web right, and so forth, of candidate and target plates.
- **Production Stage Code** - Compares the production stage code such as Stage A, and so forth of candidate and target plates.
- **Production Work Center** - Compares the production work center such as Yard1, Yard2, and so forth of candidate and target plates.
- **Number of Production Actions** - Compares the number of production actions of candidate and target plates.
- **Production Action** - Compares the routing action of candidate and target plates.
- **Type String** - Compares the type string of candidate and target plates.

Type

Select the type of check to perform.

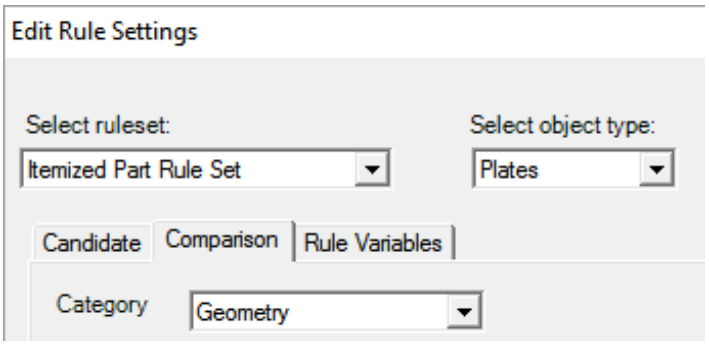
- **Attribute** - Plate properties are compared through database query.
- **Rule-based** - Plate properties are compared by plate rule.
- **Skipped** - Plate related properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Plates Type / Geometry Category Combination



Check Display Text

- **Plate Direction** - Compares the direction of the candidate and target plates.
- **Surface Area** - Compares surface area of candidate and target plates.
- **Length of Plate Boundary** - Compares outer contour length of candidate and target plates.
- **Minimum Box Enclosing Plate** - Compares minimum bounding box of candidate and target plates.
- **Plate Boundary Curve** - Compares outer contour of target and candidate plates.

Type

Select the type of check to perform.

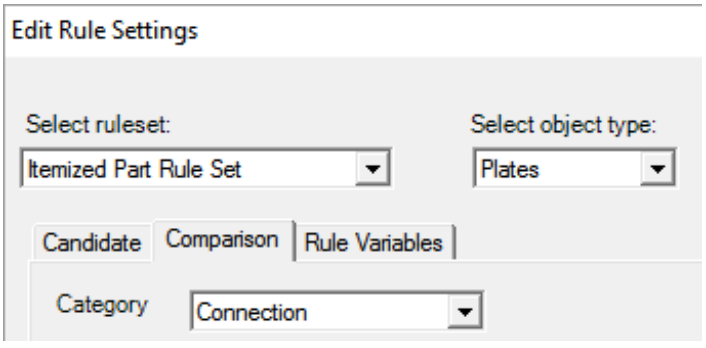
- **Rule-based** - Plate geometry properties are compared by plate rule.
- **Skipped** - Plate geometry properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Plates Type / Connection Category Combination



Check Display Text

- **Number of Physical Connections** - Compares the number of physical connections of candidate and target plates.
- **Lateral Physical Connections** - Compares the lateral physical connections of candidate and target plates.
- **Base and Offset Physical Connections** - Compares base and offset physical connections of candidate and target plates.
- **Bevels of Physical Connections** - Compares bevels of mapped connections of candidate and target plates.
- **Mounting Angles** - Compares the mounting angles of mapped connections of target and candidate plates.
- **Number of Features** - Compares the number of features of target and candidate plates.
- **Number of Edge Features** - Compares the number of edge features of target and candidate plates.
- **Number of Corner Features** - Compares the number of corner features of target and candidate plates.
- **Features** - Compares all features like sketch, edge, and corner of target and candidate plates.
- **Edge Features** - Compares edge features of target and candidate plates.
- **Corner Features** - Compares corner features of target and candidate plates.

Type

Select the type of check to perform.

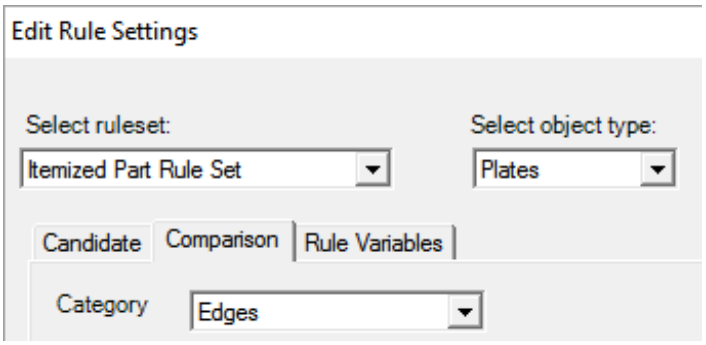
- **Attribute** - Plate connection properties are compared through database query.
- **Rule-based** - Plate connection properties are compared by plate rule.
- **Skipped** - Plate connection properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Plates Type / Edges Category Combination



Check Display Text

- **Number of Free Edge Treatments** - Compares the number of free edge treatments of candidate and target plates.
- **Free Edge Treatments** - Compares the free edge treatments of candidate and target plates.

Type

Select the type of check to perform.

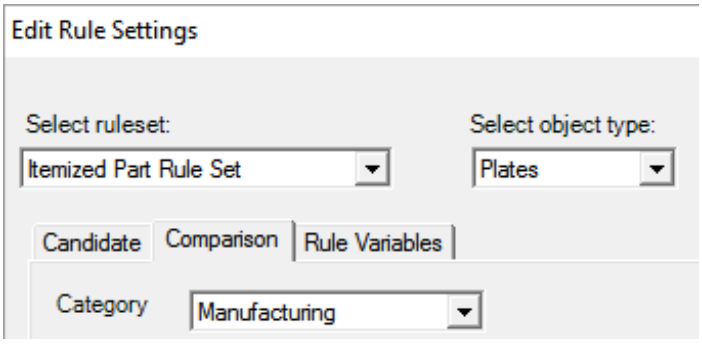
- **Attribute** - Plate edge properties are compared through database query.
- **Rule-based** - Plate edge properties are compared by plate rule.
- **Skipped** - Plate edge properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Plates Type / Manufacturing Category Combination



Check Display Text

- **Marking Lines on the Objects** - Compares the mapped pairs marking lines.

- **Shrinkage on the Objects** - Compares mapped pairs shrinkage.
- **Assembly shrinkage on the Objects** - Compares the mapped pair's assembly shrinkage.
- **Margins on the Objects** - Compares the mapped pairs margins.

Type

Select the type of check to perform.

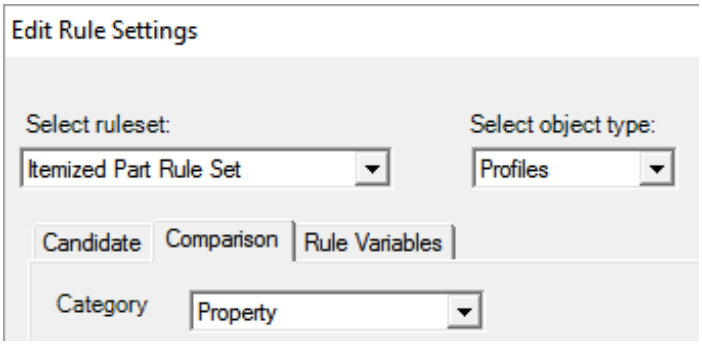
- **Rule-based** - Plate manufacturing properties are compared by plate rule.
- **Skipped** - Plate manufacturing properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Profiles Type / Property Category Combination



Check Display Text

- **Material** - Compares the material type and grade such as steel-carbon, wood, and so forth of the candidate and target profiles.
- **Definition** - Compares the cross section name (or) type of profile like B, and so forth.
- **Profile Cross Section** - Compares the profile cross section size.
- **Naming Category** - Compares the naming category like longitudinal strength, stiffener, and so forth of candidate and target profiles.
- **Profile Type** - Compares the type like longitudinal profile, transverse profile, and so forth of the profiles.
- **Profile Curvature Type** - Compares the curvature type like straight, curve, and so forth of candidate and target profiles.
- **Profile Area** - Compares the area of candidate and target profiles.
- **Profile Length** - Compares the length of candidate and target profiles.
- **Twisted** - Compares whether candidate and target profiles are twisted are not.
- **Dry Weight** - Compares the dry weight of candidate and target profiles.

- **Wet Weight** - Compares the wet weight of candidate and target profiles.
- **Building Method** - Compares the building method like slide, vertical drop, and so forth of candidate and target profiles.
- **Slot Connectivity** - Compares slot connectivity like web left, web right, and so forth of candidate and target profiles.
- **Production Stage Code** - Compares the production stage code like Stage A, and so forth of candidate and target profiles.
- **Production Work Center** - Compares production work center like Yard1, Yard2, and so forth of candidate and target profiles
- **Number of Production Actions** - Compares the number of production actions of candidate and target profiles.
- **Production Action** - Compares the routing action of candidate and target profiles.
- **Profile Dimensions** - Compares the dimensions of candidate and target profiles.
- **Surface Area** - Compares the surface area of candidate and target profiles.

Type

Select the type of check to perform.

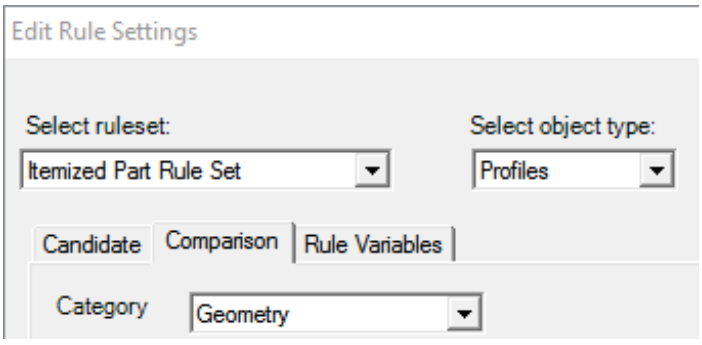
- **Attribute** - Profile properties are compared through database query.
- **Rule-based** - Profile properties are compared by plate rule.
- **Skipped** - Profile properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Profiles Type / Geometry Category Combination



Check Display Text

- **Profile Direction** - Compares the direction of candidate and target profiles.

- **Profile Primary Orientation** - Compares the primary orientation of candidate and target profiles.
- **Profile Secondary Orientation** - Compares the secondary orientation of candidate and target profiles.
- **Length of Landing Curve** - Compares the landing curve length of candidate and target profiles.
- **Minimum Box enclosing Profile** - Compares the minimum bounding box of candidate and target profiles.
- **Profile Solid Geometry** - Compares the solid geometry along with volume and movements of target and candidate profiles.

Type

Select the type of check to perform.

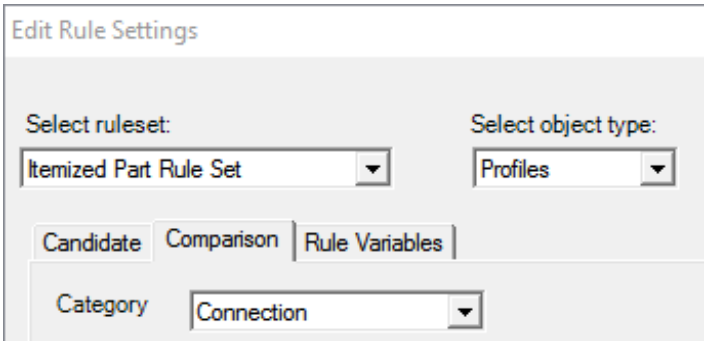
- **Rule-based** - Profile geometry properties are compared by plate rule.
- **Skipped** - Profile geometry properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Profiles Type / Connection Category Combination



Check Display Text

- **Number of Physical Connections** - Compares the number of physical connections of candidate and target profiles.
- **Lateral Physical Connections** - Compares the lateral physical connections of candidate and target profiles.
- **Base and Offset Physical Connections** - Compares the base and offset physical connections of candidate and target profiles.
- **Bevels of Physical Connection** - Compares the bevels of mapped connections of candidate and target profiles.
- **Mounting Angles** - Compares the mounting angles of mapped connections of target and candidate profiles.

- **Number of Features** - Compares the number of features of target and candidate profiles.
- **Number of Edge Features** - Compares the number of edge features of target and candidate profiles.
- **Number of Corner Features** - Compares the number of corner features of target and candidate profiles.
- **Features** - Compares all features like sketch, edge, corner features of target and candidate profiles.
- **Edge Features** - Compares edge features of target and candidate profiles.
- **Corner Features** - Compares corner features of target and candidate profiles.

Type

Select the type of check to perform.

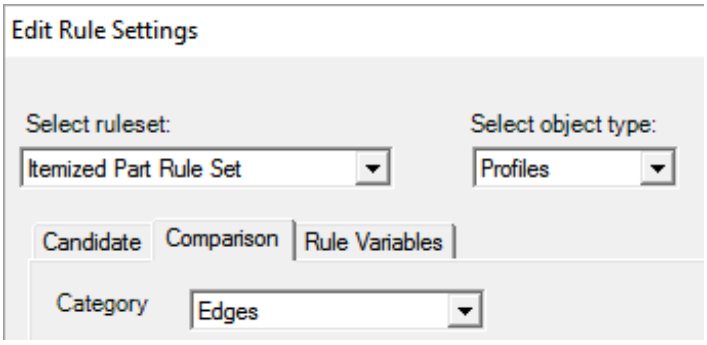
- **Attribute** - Profile connection properties are compared through database query.
- **Rule-based** - Profile connection properties are compared by plate rule.
- **Skipped** - Profile connection properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Profiles Type / Edges Category Combination



Check Display Text

- **Number of Free Edge Treatments** - compares the number of free edge treatments of candidate and target profiles.
- **Free Edge Treatments** - compares the free edge treatments of candidate and target profiles.

Type

Select the type of check to perform.

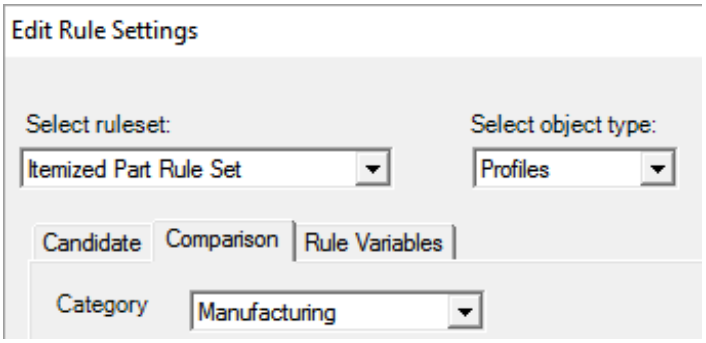
- **Attribute** - Profile edge properties are compared through database query.
- **Rule-based** - Profile edge properties are compared by plate rule.
- **Skipped** - Profile edge properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Profiles Type / Manufacturing Category Combination



Check Display Text

- **Marking Lines on the Objects** - Compares the mapped pairs marking lines.
- **Shrinkage on the Objects** - Compares the mapped pairs shrinkage.
- **Assembly shrinkage on the Objects** - Compares the mapped pair's assembly shrinkage.
- **Margins on the Objects** - Compares the mapped pairs margins.

Type

Select the type of check to perform.

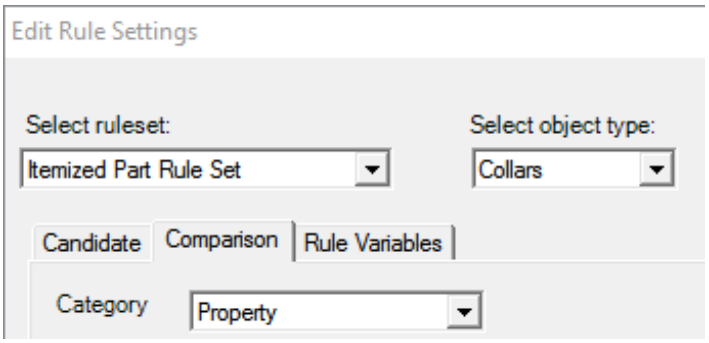
- **Rule-based** - Profile manufacturing properties are compared by plate rule.
- **Skipped** - Profile manufacturing properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Collars Type / Property Category Combination



Check Display Text

- **Collar Material** - Compares the material type and grade like steel-carbon, wood, and so forth of the candidate and target collars.
- **Collar Thickness** - Compares the thickness of candidate and target collars.
- **Collar Definition** - Compares definition like part class name (or) reference standard of collars.
- **Collar Area** - Compares the area of candidate and target collars.
- **Collar Length** - Compares the length of candidate and target collars.
- **Collar Width** - Compares the width of candidate and target collars.
- **Dry Weight** - Compares the dry weight of candidate and target collars.
- **Wet Weight** - Compares the wet weight of candidate and target collars.
- **Building Method** - Compares the build method like slide, vertical drop, and so forth of candidate and target collars.
- **Slot Connectivity** - Compares slot connectivity like web left, web right, and so forth of candidate and target collars.
- **Type String** – Compares the type string of candidate and target collars.

Type

Select the type of check to perform.

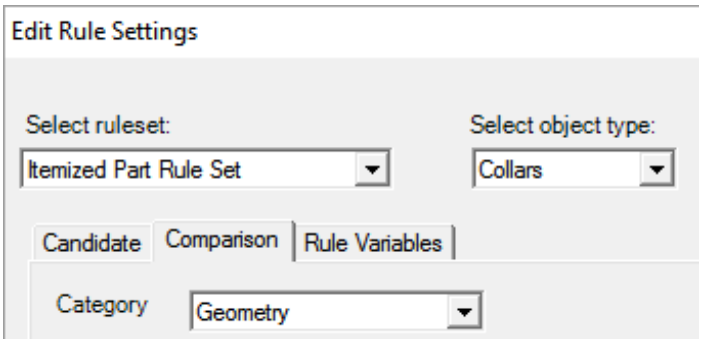
- **Attribute** - Collar properties are compared through database query.
- **Rule-based** - Collar properties are compared by plate rule.
- **Skipped** - Collar properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Collars Type / Geometry Category Combination



Check Display Text

- **Surface Area** - Compares the surface area of candidate and target collars.
- **Collar Boundary Curve** - Compares the outer contour of candidate and target collars.
- **Length of Collar Boundary** - Compares the outer contour length of candidate and target collars.

Type

Select the type of check to perform.

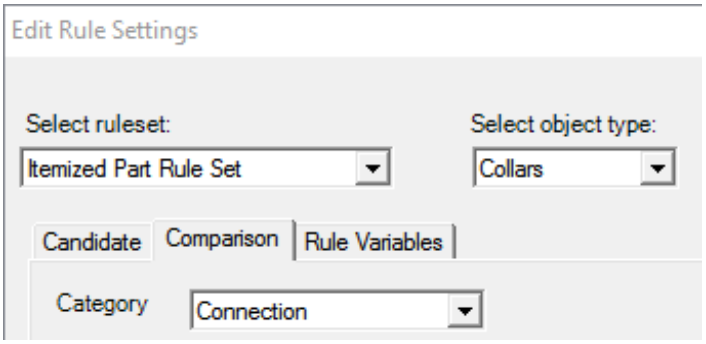
- **Rule-based** - Collar geometry properties are compared by plate rule.
- **Skipped** - Collar geometry properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Collars Type / Connection Category Combination



Check Display Text

- **Number of Physical Connections** - Compares the number of physical connections of candidate and target collars.
- **Lateral Physical Connections** - Compares the lateral physical connections of candidate and target collars.

- **Base and Offset Physical Connections** - Compares base and offset physical connections of candidate and target collars.
- **Bevels of Physical Connection** - Compares bevels of mapped connections of candidate and target collars.
- **Mounting Angles** - Compares mounting angles of mapped connections of target and candidate collars.
- **Number of Features** - Compares the number of features of target and candidate collars.
- **Number of Edge Features** - Compares the number of edge features of target and candidate collars.
- **Number of Corner Features** - Compares the number of corner features of target and candidate collars.
- **Features** - Compares all features like sketch, edge, corner features of target and candidate collars.
- **Edge Features** - Compares edge features of target and candidate collars.
- **Corner Features** - Compares corner features of target and candidate collars.

Type

Select the type of check to perform.

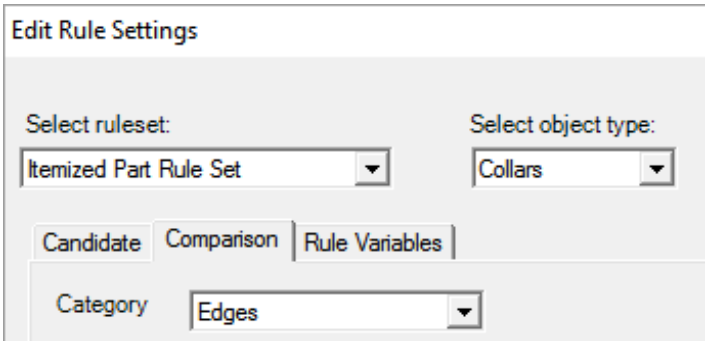
- **Attribute** - Collar connection properties are compared through database query.
- **Rule-based** - Collar connection properties are compared by plate rule.
- **Skipped** - Collar connection properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Collars Type / Edges Category Combination



Check Display Text

- **Number of Free Edge Treatments** - Compares the number of free edge treatments of candidate and target collars.
- **Free Edge Treatments** - Compares the free edge treatments of candidate and target collars.

Type

Select the type of check to perform.

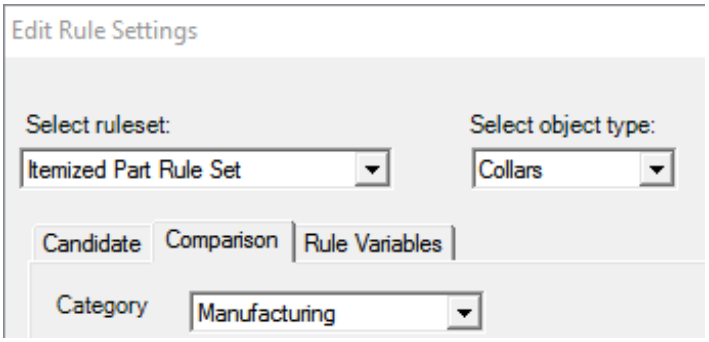
- **Attribute** - Collar edge properties are compared through database query.
- **Rule-based** - Collar edge properties are compared by plate rule.
- **Skipped** - Collar edge properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Collars Type / Manufacturing Category Combination



Check Display Text

- **Marking Lines on the Objects** - Compares the mapped pairs marking lines.
- **Shrinkage on the Objects** - Compares the mapped pairs shrinkage.
- **Assembly Shrinkage on the Objects** - Compares the mapped pair's assembly shrinkage.
- **Margins on the Objects** - Compares the mapped pairs margins.

Type

Select the type of check to perform.

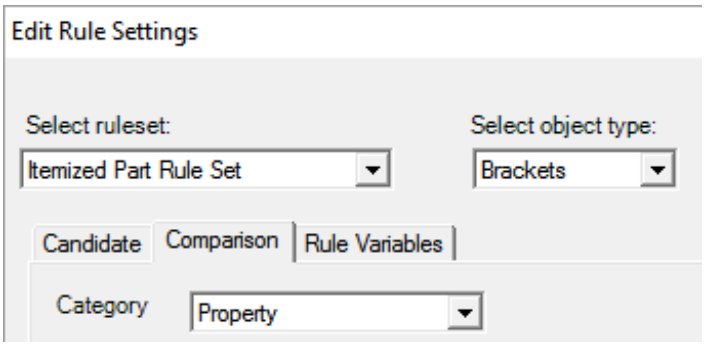
- **Rule-based** - Collar manufacturing properties are compared by plate rule.
- **Skipped** - Collar manufacturing properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Brackets Type / Property Category Combination



Check Display Text

- **Bracket Material** - Compares the material type and grade like steel-carbon, wood, and so forth of the candidate and target brackets.
- **Bracket Thickness** - Compares the thickness of candidate and target brackets.
- **Bracket Definition** - Compares the definition of brackets like part class name and reference standard.
- **Bracket Area** - Compares the area of candidate and target brackets.
- **Bracket Length** - Compares the length of candidate and target brackets.
- **Bracket Width** - Compares the width of candidate and target brackets.
- **Dry Weight** - Compares the dry weight of candidate and target brackets.
- **Wet Weight** - Compares the wet weight of candidate and target brackets.
- **Building Method** - Compares the build method like slide, vertical drop, and so forth of candidate and target brackets.
- **Slot Connectivity** - Compares slot connectivity like web left, web right, and so forth of candidate and target brackets.
- **Type String** - Compares the type string of candidate and target brackets.

Type

Select the type of check to perform.

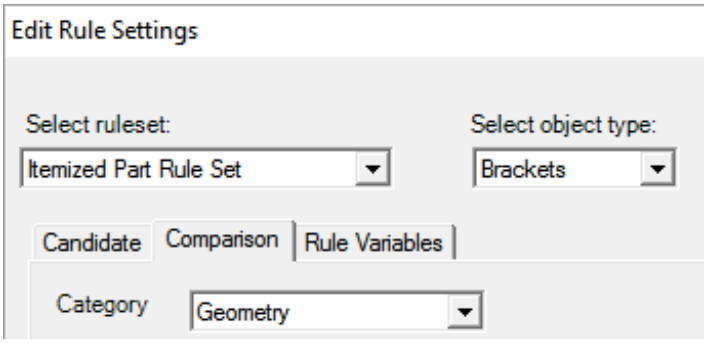
- **Attribute** - Bracket properties are compared through database query.
- **Rule-based** - Bracket properties are compared by plate rule.
- **Skipped** - Bracket properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Bracket Type / Geometry Category Combination



Check Display Text

- **Surface Area** - Compares the surface area of candidate and target brackets.
- **Bracket Boundary Curve** - Compares the outer contour of candidate and target brackets.
- **Length of Bracket Boundary** - Compares the outer contour length of candidate and target brackets.

Type

Select the type of check to perform.

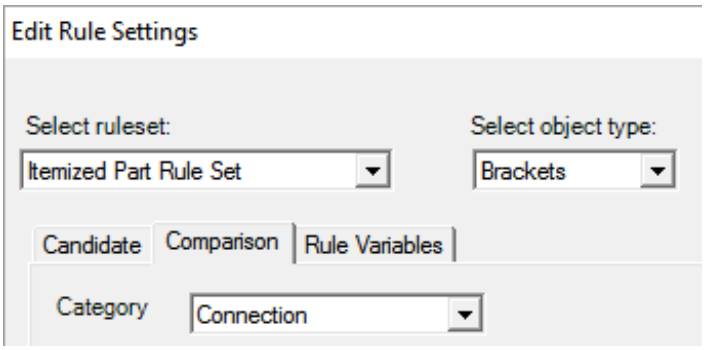
- **Rule-based** - Bracket geometry properties are compared by plate rule.
- **Skipped** - Bracket geometry properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Bracket Type / Connection Category Combination



Check Display Text

- **Number of Physical Connections** - Compares the number of physical connections of candidate and target brackets.
- **Lateral Physical Connections** - Compares the lateral physical connections of candidate and target brackets.
- **Base and Offset Physical Connections** - Compares the base and offset physical connections of candidate and target brackets.
- **Bevels of Physical Connection** - Compares bevels of mapped connections of candidate and target brackets.
- **Mounting Angles** - Compares mounting angles of mapped connections of target and candidate brackets.
- **Number of Features** - Compares the number of features of target and candidate brackets.
- **Number of Edge Features** - Compares the number of edge features of target and candidate brackets.
- **Number of Corner Features** - Compares the number of corner features of target and candidate brackets.
- **Features** - Compares all features like sketch, edge, corner of target and candidate brackets.
- **Edge Features** - Compares the edge features of target and candidate brackets.
- **Corner Features** - Compares the corner features of target and candidate brackets.

Type

Select the type of check to perform.

- **Attribute** - Bracket connection properties are compared through database query.
- **Rule-based** - Bracket connection properties are compared by plate rule.
- **Skipped** - Bracket connection properties are not compared.

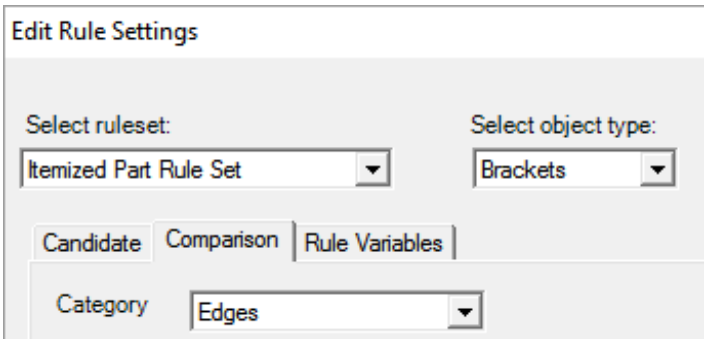
Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Brackets Type / Edges Category Combination



Check Display Text

- **Number of Free Edge Treatments** - Compares the number of free edge treatments of candidate and target brackets.
- **Free Edge Treatments** - Compares the free edge treatments of candidate and target brackets.

Type

Select the type of check to perform.

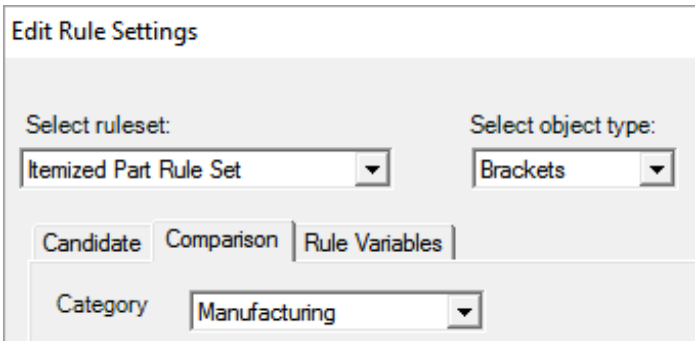
- **Attribute** - Bracket edge properties are compared through database query.
- **Rule-based** - Bracket edge properties are compared by plate rule.
- **Skipped** - Bracket edge properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Brackets Type / Manufacturing Category Combination



Check Display Text

- **Marking Lines on the Objects** - Compares the mapped pairs marking lines.
- **Shrinkage on the Objects** - Compares mapped pairs shrinkage.
- **Assembly shrinkage on the Objects** - Compares the mapped pair's assembly shrinkage.
- **Margins on the Objects** - Compares the mapped pairs margins.

Type

Select the type of check to perform.

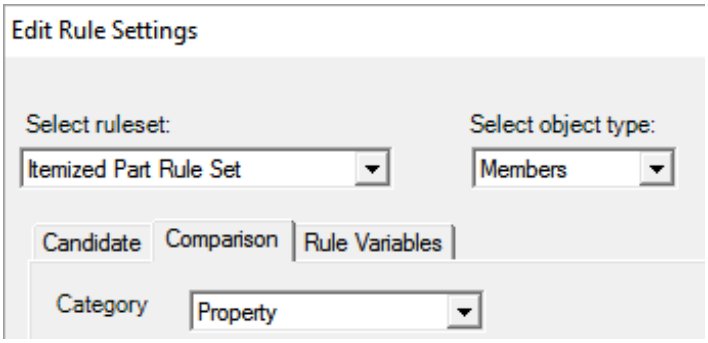
- **Rule-based** - Bracket manufacturing properties are compared by plate rule.
- **Skipped** - Bracket manufacturing properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Members Type / Property Category Combination



Check Display Text

- **Material** - Compares the material type and grade like steel, wood, and so forth of the candidate and target members.

- **Definition-Section** - Compares the section type like 2L, C, and so forth of candidate and target members.
- **Definition-Attributes** - Compares the attributes of candidate and target members.
- **Definition-Section Name** - Compares the section name like 2L7X4X3/8LLBB, 2L6X4X3/4LLBB, and so forth of candidate and target members. If the section selected is 2L.
- **Definition-Section Standard** - Compares the section standard like AISC-LRFD-3.1, and so forth of candidate and target members.
- **Cut Length** - Compares the cut length of candidate and target members.
- **Type Category** - Compares the type category like Beam, Column, and so forth of candidate and target members.
- **Type** - Compares the type like Beam, Girder, and so forth of candidate and target members.
- **Priority** - Compares the priority like primary, secondary, and so forth of candidate and target members.
- **Length** – Compares the curve length of candidate and target members.
- **Dry Weight** - Compares the dry weight of candidate and target members.
- **Wet Weight** - Compares the wet weight of candidate and target members.
- **Coating Requirements** - Compares the coating requirements of candidate and target members.
- **Coating Type** - Compares the coating type of candidate and target members.
- **Coating Color** - Compares the coating color of candidate and target members.
- **Coating Area** - Compares the coating area of candidate and target members.
- **Construction Requirement** - Compares the construction requirement of candidate and target members like new, existing, and so forth.
- **Construction Type** - Compares the construction type of candidate and target members.
- **Fabrication Requirement** - Compares the fabrication requirement of candidate and target members like by fabricator, by erector, and so forth.
- **Fabrication Type** - Compares the fabrication type of candidate and target members.
- **Reporting Requirement** - Compares the reporting requirement of candidate and target members like to be reported or not to be reported.
- **Reporting Type** - Compares the reporting type of candidate and target members.
- **Cleaning Responsibility** - Compares the cleaning responsibility of candidate and target members like By Piping, By Instruments, and so forth.
- **Design Responsibility** - Compares the design responsibility of candidate and target members like By Piping, By Instruments, and so forth.
- **Fabrication Responsibility** - Compares the fabrication responsibility of candidate and target members like By Piping, By Instruments, and so forth.

- **Painting Responsibility** - Compares the cleaning responsibility of candidate and target members like By Piping, By Instruments, and so forth.
- **Requisition Responsibility** - Compares the requisition responsibility of candidate and target members like By Piping, By Instruments, and so forth.
- **Supply Responsibility** - Compares the supply responsibility of candidate and target members like By Piping, By Instruments, and so forth.
- **Testing Responsibility** - Compares the testing responsibility of candidate and target members like By Piping, By Instruments, and so forth.
- **Piece Mark** - Compares the piece mark of candidate and target members.
- **Assembly Mark** - Compares the assembly mark of candidate and target members.
- **Production Stage Code** - Compares the production stage code candidate and target members.
- **Production Work Center** - Compares the production work center of candidate and target members.
- **Number of Production Actions** - Compares the number of production actions of candidate and target members.

Type

Select the type of check to perform.

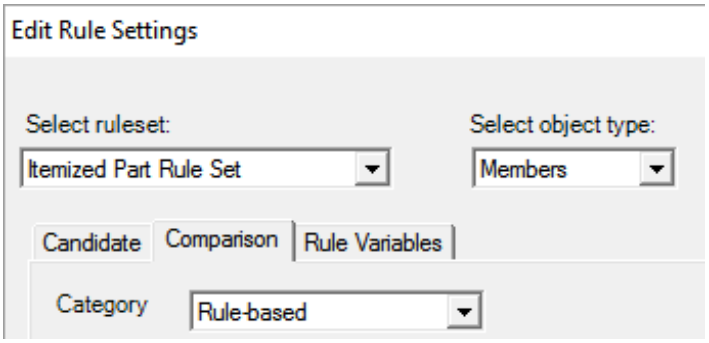
- **Attribute** - Member properties are compared through database query.
- **Rule-based** - Member properties are compared by plate rule.
- **Skipped** - Member properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Members Type / Rule-based Category Combination



Check Display Text

- **Production Action** - Compares the routing action of candidate and target members.

Type

Select the type of check to perform.

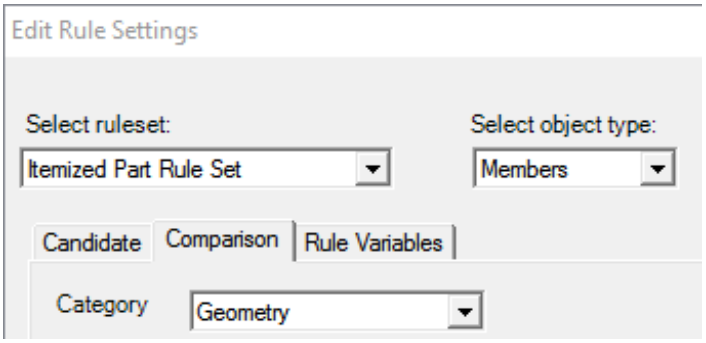
- **Rule-based** - Member rule-based related properties are compared by plate rule.
- **Skipped** - Member rule-based related properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Members Type / Geometry Category Combination



Check Display Text

- **Member Direction** - Compares the direction of candidate and target members.
- **Geometry** - Compares the geometry of candidate and target members.

Type

Select the type of check to perform.

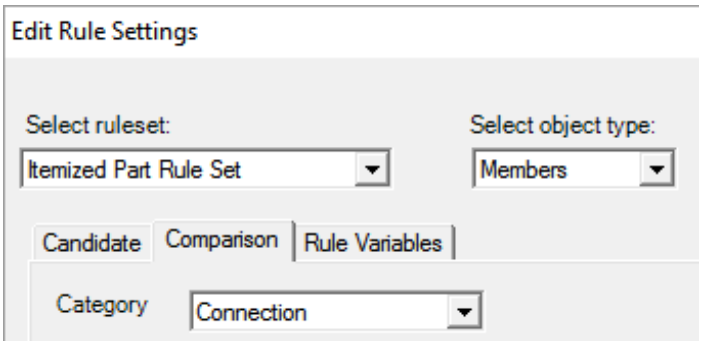
- **Rule-based** - Member geometry properties are compared by plate rule.
- **Skipped** - Member geometry properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Members Type / Connection Category Combination



Check Display Text

- **Number of Physical Connections** - Compares the number of physical connections of candidate and target members.
- **Bevels of Physical Connection** - Compares bevels of physical connections of candidate and target members.
- **Bevel and Mounting Angles** - Compares mounting angles of mapped connections of target and candidate members.

Type

Select the type of check to perform.

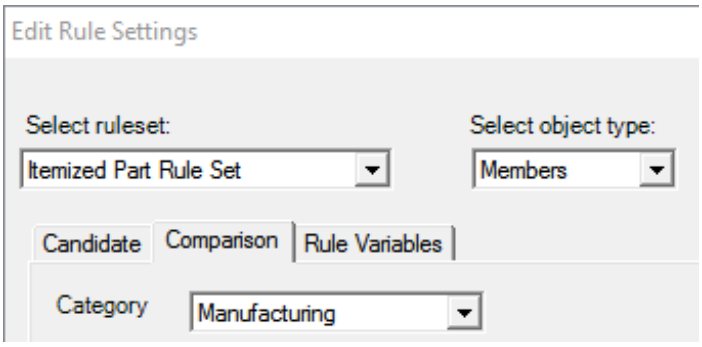
- **Attribute** - Member connection properties are compared through database query.
- **Rule-based** - Member connection properties are compared by plate rule.
- **Skipped** - Member connection properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Attribute**, click **View/Edit** to define **Select** and **Join** information.
- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Members Type / Manufacturing Category Combination



Check Display Text

- **Marking Lines on the Objects** - Compares the mapped pairs marking lines.
- **Shrinkage on the Objects** - Compares the mapped pairs shrinkage.
- **Assembly shrinkage on the Objects** - Compares the mapped pair's assembly shrinkage.
- **Margins on the Objects** - Compares the mapped pairs margins.

Type

Select the type of check to perform.

- **Rule-based** - Member manufacturing properties are compared by plate rule.
- **Skipped** - Member manufacturing properties are not compared.

Value

Defines the comparison value.

- If **Type** is set to **Rule-based**, the appropriate rule prog-id is displayed. You can edit this prog-id if needed.
- If **Type** is set to **Skipped**, the **Value** field is disabled.

Rule Variables (Edit Rule Settings Dialog Box)

Variables

Specifies the name of the rule variable.

- **Distance Tolerance** - Compares distance related properties like thickness, width, and so forth (applicable for Plates, Profiles, Collars, Brackets and Members.)
- **Area Tolerance** - Compares area related properties like area, surface area, and so forth (applicable for Plates, Profiles, Collars, Brackets and Members.)
- **Volume Tolerance** - Compares volume related properties like bounding box, and so forth (applicable for Plates, Profiles, Collars, Brackets and Members.)
- **Default Tolerance** - Compares properties like length, and so forth (applicable for Plates, Profiles, Collars, Brackets and Members.)
- **Weight Tolerance** - Compares properties like dry weight, and so forth (applicable for Plates, Profiles, Collars, Brackets and Members.)
- **Flipping Part** - Checks whether part is flipped or not. (applicable for Plates, Collars and Brackets)
- **Bevel Validation Level** - If **Bevel Validation Level** is 2, then it compares common properties and it checks whether part is flipped or not. If Flipped "Ref-AntiRef" (or) "AntiRef-Ref" comparison is done. If it's not flipped "ref-ref" (or) "antiref-antiref" comparison is done. If **Bevel Validation Level** is other than 2, then only common properties are compared (applicable for Plates, Profiles, Collars, Brackets and Members.)
- **Angle Tolerance** - Compares angle related properties like bevel comparison, and so forth (applicable for Plates, Profiles, Collars, Brackets and Members.)

Units

Indicates the unit used to specify the variable value.

Values

Specifies the value of the variable.

Start Test Command

Initiates the **Test Rules** command.

Save As

Prompts you to name the rule settings and save them in a required location.

Save

Saves the rule setting in the current location.

Close

Closes the **Edit Rule Setting** dialog box.

Edit the rule settings

1. Click **Tools** > **Common Part Utilities** > **Edit Rule Settings**.

   *The **Edit Rule Setting** dialog box is displayed.*
2. Select the required rule set in the **Select ruleset** list.
3. Select the required object typed in the **Select object type** list.

   *The **XML Path** is updated.*
4. Select the required tab and edit the rule settings as required.

SECTION 22

Planning Utilities

Planning Utilities are used to perform tasks such as testing block assignment, removing assembly children, and repairing orphan objects. **Planning Utilities** is located on the **Tools** menu.

Repair Planning Objects

Repairs the following Planning objects:

- Assemblies without a parent
- Parts with multiple parents
- Assemblies with an invalid status
- Assemblies in a hierarchical loop
- Second-class business objects with an assembly hierarchy relation
- Blocks with invalid geometry
- Common part groups without a common entity master relationship

Repair Planning objects

1. In the Planning task, click **Tools > Planning Utilities > Repair Planning Objects**.

   *The **Repair Planning Objects** dialog box displays.*

2. Use **Select Options** to specify the Planning object type to repair.

   The software populates the dialog box with the appropriate options.

3. Select the remaining options as needed. For more information, see *Repair Planning Objects Dialog Box* (on page 197).

Repair Planning Objects Dialog Box

Displays options for repairing Planning objects.

Select Options

Specifies the type of planning object to repair. The object type that you select controls the content that displays in the dialog box. Select one of the following: **Orphan assemblies**, **Parts with multiple parents**, **Blocks with invalid geometry**, **Invalid common part groups**, **Assemblies in loop**, **Assemblies with invalid status**, or **Non-FCBOs in hierarchy**.

Orphan Assemblies Options

Orphan Assemblies (#children)

Lists the assemblies or assembly blocks without a parent.

Parent

Specifies the parent assembly or block. To view more options, **More** to open the **Select Assembly** dialog box.

Assign

Assigns the specified parent to the orphan assemblies selected in the **Orphan Assemblies (#children)** list.

Delete

Deletes the selected orphan assemblies.

Parts with Multiple Parents Options

Parts with multiple parents

List the parts that have multiple parents.

Parent

Specifies the parent assembly or block. To view more options, **More** to open the **Select Assembly** dialog box.

Assign

Assigns the specified parent to the part that you selected in the **Parts with multiple parents** list.

Blocks with Invalid Geometry Options

Blocks with invalid geometry

Lists the blocks with invalid geometry.

Delete

Deletes the selected block.

Assemblies in Loop Options

Assemblies in loop

List the assemblies or assembly blocks in a hierarchical loop.

Parent

Specifies the parent assembly or block. To view more options, **More** to open the **Select Assembly** dialog box.

Assign

Assigns the specified parent to the assemblies in loop that you selected in the **Assemblies in loop** list.

Delete

Deletes the selected assemblies in loop.

Assemblies with Invalid Status

Assemblies with invalid status

Lists the assemblies or assembly blocks that have an invalid status.

Status

Specifies the current status. Available options are **Working**, **In Review**, **Rejected**, and **Approved**.

Assign

Assigns the specified status to the assemblies or assembly blocks that you selected in the **Assemblies with invalid status** list.

Non-FCBOs in Hierarchy Options

Non-FCBOs in hierarchy

Lists second class business objects that have an assembly hierarchy relationship.

Break Hierarchy

Breaks the assembly hierarchy relationship of the second class business object selected in the list.

Invalid Common Part Groups Options

Invalid common part groups

Lists the common part groups without a master part or CommonEntityMaster relation.

Select Master (Optional)

Lists the parts that belong to the selected common part group. When you select a part in the list, the software highlights the object in the **Workspace Explorer**.

NOTE By default, the software highlights any part that has a relation name. This indicates that the part was a master part in a previous version of the software.

Set Master Automatically

Assigns the master automatically to all of the listed common part groups.

Set Master

Assigns the part selected in the **Select Master** list as the master to the selected common part group. This option is not available if you select more than one common part group.

Test Block Assignment

Allows you to test block assignment using one of the following user-specified methods:

- Run the block assignment process on a group of selected parts.
- Run the block assignment process on a single model object.
- Run the block assignment process on model data uploaded to the catalog in an Excel worksheet.

The block assignment process assigns and reassigns parts to blocks. For more information, see *Block Assignment* (on page 156).

Test block assignment

1. In the Planning task, click **Tools > Planning Utilities > Test Block Assignment**.

   *The **Test Block Assignment** dialog box displays.*

2. Select the **User Options** as needed, and then click **Run**.

   *The software runs the block assignment process and assigns the parts to the block as specified in **User Options**.*

NOTES

- Before selecting **Use Select Set**, you must first select the parts on which to run the block assignment in the **Workspace Explorer**.
- If you select **Use Excel Data**, you must specify the full path location and file name of the Excel file to upload in the **Excel** box.

Test Block Assignment Dialog Box

Specifies options for testing block assignment.

Use Select Set

Runs the block assignment process on a user-specified group of parts. You must select the parts in the **Workspace Explorer** before you use the **Test Block Assignment** command.

Name/OID

Specifies a single part to be assigned to the block. Type the name or OID of the model object.

Use Excel Data

Runs the block assignment process using the data in the .xls file specified in the **Excel** box. An example of sample data is shown below:

	A
1	<NoBlk>-LS.1-1
2	<NoBlk>-LS.11-1
3	0010A20F-0000-0000-2616-4A32D9561D04
4	0010A20F-0000-0000-6D2B-4A32D9561D04
5	

Excel

Specifies the full path location and file name of the Excel worksheet to upload. Click the

browse button  , and navigate to the required file. You must define this option if you select **Use Excel Data**.

Run

Starts the block assignment process.

SECTION 23

Check Manufacturability

Tools > **Check Manufacturability** (**Tools > Check Hole** in the **Hole Management** task) analyzes objects in the current workspace and reports the objects that will be difficult or impossible to manufacture according to standards defined in the reference data.

Check Manufacturability Ribbon

Provides the following options to check that objects in the model can be manufactured and to generate production information.



Manufacturability Checking Settings

Shows the rules for the checking process. You can select any or all of the rules. For more information, see *Manufacturability Checking Settings Dialog Box* (on page 204).



Check Manufacturability

Starts the checking process. If inconsistencies are found, the message **Manufacturability errors/warnings have been encountered** displays.



Show Manufacturability Inconsistencies

Displays the objects with manufacturing issues. The list includes the issue severity, the assembly name, a description of the assembly, and the name of the rule which detected the manufacturing issue. For more information, see *Show Manufacturability Inconsistencies Dialog Box* (on page 205).

Close

Exits the command.

Submit Job

Displays a message box asking if you want to execute the repair rules subsequently. If you click **Yes**, the **Schedule Planning Check Manufacturability** dialog box displays so that you can schedule the execution of the repair rules at some other time. For more information, see *Schedule [Task] Dialog Box (on page 160)* in the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

NOTES

- Learn more about SmartPlant Batch Services in the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.
- The **Submit Job** button is enabled only when Batch Services is installed and started on your computer.
- The data on objects that cannot be manufactured is stored in the session file, not in the model. Problem objects that apply to the Workspace are retrieved when you select the **Check Manufacturability** command.

What do you want to do?

- *Check objects for manufacturability* (on page 203)
- *Schedule check manufacturability using Batch Services* (on page 203)
- View the *Manufacturability Checking Settings Dialog Box* (on page 204)
- View the *Show Manufacturability Inconsistencies Dialog Box* (on page 205)

Check objects for manufacturability

1. Select objects for checking in the model or in the **Workspace Explorer**.
2. Click **Tools > Check Manufacturability** (**Tools > Check Holes** in Hole Management).
3. On the ribbon, click **Manufacturability Checking Settings** .
4. Specify the rules for the checking process.
5. On the ribbon, click **Check Manufacturability** to start the process.
6. On the ribbon, click **Show Manufacturability Inconsistencies** .

   A list view displays the objects that were found to have manufacturing problems.
7. Select one row of the list at a time to view the inconsistency and solution for an object.
8. Repair the object manually, or click **Repair** if an **Action Tool** is available.

Schedule check manufacturability using Batch Services

1. Select objects for checking in the model or in the **Workspace Explorer**.
2. Click **Tools > Check Manufacturability**.
3. On the ribbon, click **Manufacturability Checking Settings** .
4. Specify the rules for the checking process.
5. Click **Submit Job**.
6. Click **Yes** to schedule the job for another time.
7. Complete the **Schedule Check Manufacturability** dialog box to configure batch processing, and then click **OK**.

   The software displays the message, "This job has been scheduled to run on the chosen server."
8. Click **OK** to dismiss the message.

NOTES

- For more information about SmartPlant Batch Services, see the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

- After the job is successfully submitted, you can see the PlanningCheckMfcty_BatchJob.xml file in the system temp folder (%temp%). This file contains the required information to run the batch job.
- If any inconsistencies are observed during the execution of the check manufacturability rules that you selected, information about the name of the object, manufacturability rule ProgID, and repair rule ProgID are written to the PlanningCheckMfcty_InConsistencies.log file in the system temp folder (%temp%).

Manufacturability Checking Settings Dialog Box

Sets options for the **Check Manufacturability** process.

Task List

Displays the list of tasks that contains the **Check Manufacturability** rules defined in the catalog. These tasks allow you to filter the check rules based on the selected task in the **Task List**.

Rule

Select which rules to process check manufacturability against. There are different rules based on the task selected in the **Task List** option. Contact Intergraph Support http://support.intergraph.com for help with customizing the XML data file.

- The **Hole Management** manufacturability checking process includes checking the distance between hole traces; the distance between the hole trace and plate edges (outside), brackets and coamings; and the distance between the hole trace and seamlines, profiles, and openings. Because hole traces can affect the stress of the deck or bulkhead on which they are placed, it is important to check holes before cutting them. For more information about hole management checks, see Check Hole Rules in the *Hole Management Reference Data Guide*.
- The **Piping** manufacturability checking process includes software simulators for cutting, bending, and coating pipe. An XML data file controls the piping simulators. You can customize this file to reflect the requirements, standards, and equipment of your facility. For more information about piping checks, see Piping Check Manufacturability in the *Piping Reference Data Guide*.
- The **Planning** manufacturability checking process includes software simulators for planning joints, weld bevels, and slot types on structure. You can customize the planning simulators rules to reflect the requirements, standards, and equipment of your facility. For more information about planning checks, see Check Manufacturability Rules in *Planning Reference Data*.
- The **Structural Detailing** checking process includes tee weld chamfer checks based on changes to plate thickness, slot opening angles, and end cut type. For more information, see Check Manufacturability Rules in *Structural Detailing Reference Data*.
- The **Structural Manufacturing** checking process includes margin check on the same port, manufacturing plat and profile check, and shrinkage check.

ProgID

Programming ID of the rule that found the inconsistency. Matches the **Rule ProgID** of the rule on the *Show Manufacturability Inconsistencies Dialog Box* (on page 205)  .

Select All

Specifies all available rules.

Clear All

Clears all available rules. You can select rules individually in the list view by clicking the boxes beside the rule names.

Stop checking if a manufacturability error is found

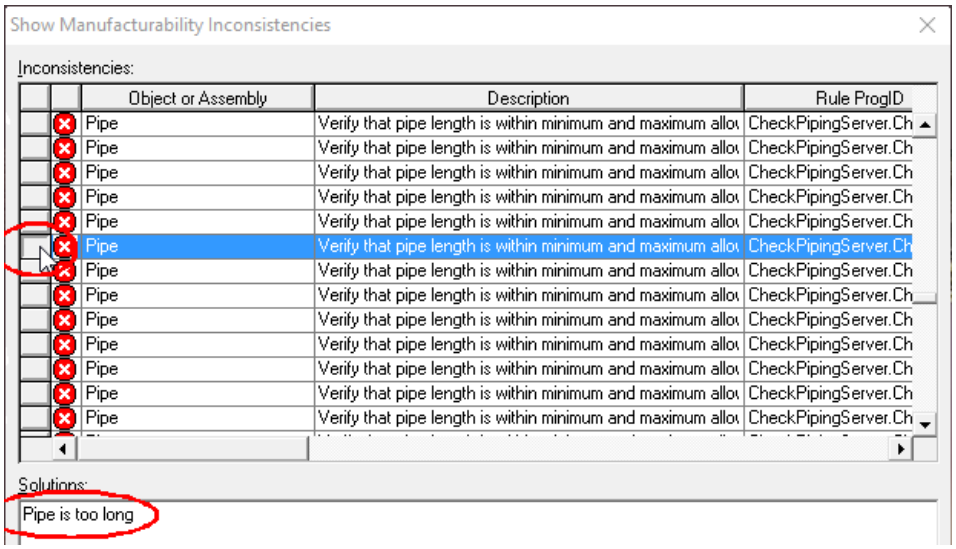
Halts rule processing upon error detection.

Show Manufacturability Inconsistencies Dialog Box

Displays and repairs inconsistencies found by the rules of the **Check Manufacturability** process.

Inconsistencies

A list view of all inconsistencies. Click the button in the far left column to see the possible solution. Solution descriptions are listed below.



Warning

A minor manufacturability problem has been found. The problem does not have to be repaired.

Error

A major manufacturability error has been found. The error must be repaired.

Object or Assembly

Name of the object with an inconsistency.

Description

Description from the rule of the inconsistency

Rule ProgID

Programming ID of the rule that found the inconsistency. Matches the **ProgID** of the rule on the **Manufacturability Checking Settings Dialog Box** (on page 204)  .

Action

The type of action that is available in the software to repair the inconsistency. Blank if no software action is available and the repair is done manually.

- **COM Repair Object** - Software code that can be run.
- **SQL Script** - SQL Script that can be run.

Action Tool

ProgID of the COM Repair Object or name of the SQL Script. Blank if **Action** is blank.

Solutions

Describes the details of an inconsistency and the repair that you need to perform for one selected object from the list.

Repair

Runs the **Action Tool** to perform a repair, if one is available for the rule.

Fit

Fits one or more selected objects from the list view in the active graphic view.

Clear

Clears all inconsistencies and closes the dialog box. You must click **Check Manufacturability**  again to display the remaining inconsistencies.

Piping Inconsistencies Solution Messages

Pipe and Spool Length Messages

Pipe is too long

Pipe exceeds standard ordering length for pipe of this size, as defined in the pipe catalog. For bent pipes the length check includes any extra pipe that had to be added at the ends or between bends for clamping (see "Extra pipe must be added at end…" and "Insufficient straight pipe between bends…" messages below.). If the pipe is bent, the elongation of the pipe during bending will be taken into account when reporting this error.

Pipe is too short

Pipe length is less than the company standard for pipes welded on both ends.

Spool is too long

The overall length of the spool exceeds company standards. The length is measured along the main axis of the spool.

Spool is too wide

The "width" in the spool is measured perpendicular to the main axis. This message indicates that the width exceeds company standards. Checks for surface treatment tank size are done separately.

Pipe Bending Messages

Bend angle too large

A pipe bend exceeds the maximum bend angle that the bending machine can make.

Bend angle too small

A pipe bend angle is less than the company standard for bends. That is, the pipe is almost straight.

Bends have different radii, not allowed

Pipe has two or more bends, and they do not all have the same bend radius specified.

Bend radius too large or points too close together

The pipe geometry is physically impossible. This is an extreme case of the "insufficient straight pipe between bends…" problem. This problem may be fixed by moving a bend or using a smaller bend radius.

Extra pipe must be added at end, pipe end too short for bending machine to clamp onto

The bending machine needs a sufficiently long straight section at the start and end of the pipe to clamp onto (start), and support (end). This is a warning; bending can be done with a longer piece of pipe, with the extra pipe cut off afterwards.

Insufficient straight pipe between bends for bending machine to clamp onto

The bending machine needs a sufficiently long, straight section between bends to clamp the pipe. This problem may be fixed by moving a bend, by using a smaller bend radius, or by splitting the pipe and putting a joint between the bends.

No pipe bending machine for pipe diameter and bend radius

None of the pipe bending machines listed in the pipe shop XML data file have bend dies listed for the pipe diameter and bend radius of this pipe.

Pipe hits machine or floor during bending

The pipe cannot be bent because the free end or part of the pipe would hit the bending machine or the shop floor during bending. The simulator will have evaluated bending the pipe starting from either end before reporting this error, and will have checked all available bending machines with bend dies of the right size.

NOTE If the pipe has flanges on either end, and the **Flange Welding** check option has been selected, the simulator also checks for attached flange (if any) hitting the machine during bending.

Pipe and Spool Complexity

Too many bends in pipe

The number of bends in the pipe exceeds the company standard. This limit is normally set to maintain dimensional accuracy standards because of accumulated error during bending.

Too many branches

The number of branches off of a main exceeds the company standard limit.

Too many planes in spool

The spool is difficult to fabricate because it is geometrically complex. That is, the number of independent planes exceeds the company standard for spools.

Treatment Tank

Spool too large for treatment tank

A spool that requires galvanization or other tank treatment is too large for the tank.

No treatment tank found for <treatment_name> required by pipe spool

No treatment tank has been given in PipeBenders.xml for the treatment type called out by the pipe specification. This is an error in the editing of the PipeBenders.xml file, not a design error in the piping.

Individual pipes in spool require different tank treatments

A pipe spool is composed of multiple pipes that reference different pipe specs, and those pipe specs call out different, conflicting tank treatments.

Flange Welding

Flange hits machine or floor during bending, weld after bending

Flanges cannot be welded on the pipe before bending because they would hit the bending machine or the shop floor during bending. The simulator will have evaluated bending the pipe starting from either end before reporting this error, and will have checked all available bending machines with bend dies of the right size.

Pipe too long for automatic flange welding

This is a warning that a straight pipe with flanges on one or both ends is too long to be put through the automatic flange welder, and must be manually welded.

Pipe too short for automatic flange welding

This is a warning that a straight pipe with flanges on one or both ends is too short to be put through the automatic flange welder, and must be manually welded.

Inside Grinding and Painting

Pipe too long for inside coating

A pipe requires internal coating, but the pipe is too long for the available Pipe Internal Sprayer.

No internal sprayer found for inside coating

A pipe requires internal coating (as defined in the pipe spec entry in the XML file), but there is no Pipe Internal Sprayer available for the required coating material or for the pipe diameter.

Inaccessible for internal coating

A pipe that requires internal coating after bending has two or more bends. The sections between the bends are inaccessible for coating.

Bend on branch not allowed

A branch weld is inaccessible for grinding after welding because of a bend in the branch pipe.

Branch too far from end of main

Grinding cannot be done on a branch connection because it is too far from the end of the pipe.

Planning Inconsistencies Solution Messages

First Meet Check

First Meet assembly does not match the Planning Joint assembly

The planning joint is not located under the assembly containing the parts joined by the planning joint.

Hierarchy Check

Planning joint is not in an assembly that contains the two joined parts

The planning joint is located neither under the assembly containing the parts joined by the planning joint nor under a parent assembly/block of the assembly.

Physical Connection Check

Weld name is unknown

The weld bevel of a physical connection is not known because of a change to its associated planning joint.

Production Equipment Check

Production Equipment is not assigned to the assembly

The assembly has no production equipment assigned or the assigned production equipment is not compatible and cannot perform the weld.

Weld Side Check

The Weld Side of the Planning Joint is incorrect

The weld side of the planning joint is incorrect.

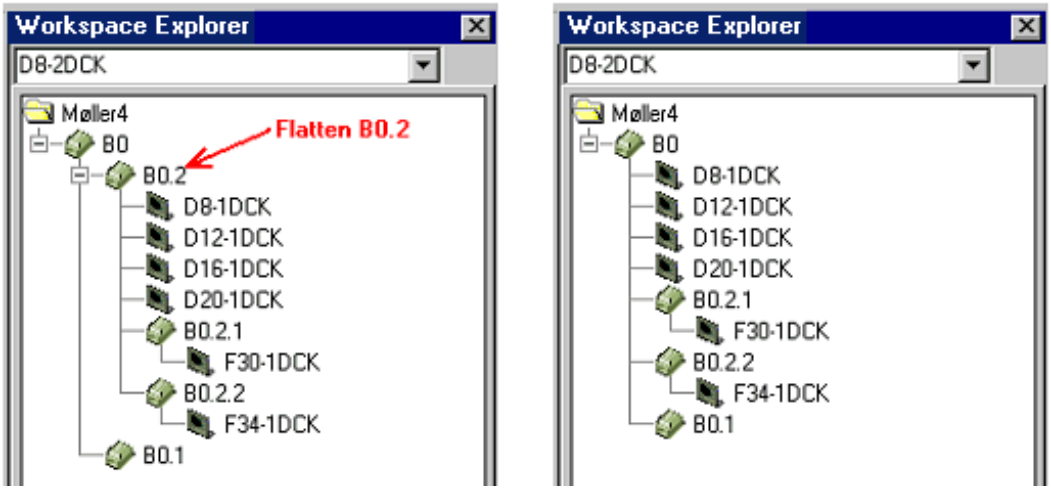
SECTION 24

Flatten Block/Assembly/Assembly Block

Flattens assemblies, blocks, and assembly blocks. The command performs the function of "deleting" an assembly or blocks without deleting its contents. The node of the selected assembly or block is removed from the assembly hierarchy in the **Workspace Explorer**. All immediate children of the selected block or assembly take the place of the "deleted" assembly or block with the parent. As a result, unwanted intermediate levels of the assembly hierarchy are removed. All the children of the "flattened" assembly or block become children of its parent. For blocks, their children are in effect *nested* into the volume of the parent block because they occupy part of the volume of the flattened block itself.

This command is available only on the right-click shortcut menu in the **Workspace Explorer**.

In the following example, block B0.2 is flattened; the resulting hierarchy is shown on the right:



Flatten an assembly or block

1. Select the assembly, block, or assembly block to flatten in the **Workspace Explorer.**
2. Right-click, and select **Flatten Block/Assembly/Assembly Block** on the shortcut menu.

TIP The name of the command changes to reflect the type of Planning item that you select.

NOTES

- The **Flatten** command is only available on the shortcut menu of the selected assembly, block, or assembly block.
- The **Flatten Block** command is not available for the top node block of the Model and for bottom leaf level blocks. Bottom leaf blocks cannot be because all the volume of higher-level blocks must be assigned to lower level blocks.

Glossary

abaft

Toward the stern of a ship, behind, further aft than.

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

aft

Toward, at, or near the stern.

after body

The hull from aft of the midship section.

aftermost

Nearest the stern.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

assembly

Collection of assemblies, structural parts, equipment, outfitting parts, and outfitting assemblies (such as pipe spools). An assembly is a child of a block, assembly block, or another assembly.

assembly block

Collection of blocks, assemblies, structural parts, equipment, outfitting parts, and outfitting assemblies (such as pipe spools). An assembly block is a child of the top-level block or another assembly block.

assembly process

Processes and sequences used to manufacture and assemble parts into block units.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

auto-nesting

Automatic selection and location of various shapes and sizes of steel plate parts to fit on a standard steel plate, minimizing scrap while optimizing cutting time and minimizing thermal distortion.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

basic design

Engineering definition of the model and its systems.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

block

High-level assembly that is a spatial definition of a section of the marine structure. Blocks are created by splitting a volume and contain assemblies, parts, and/or child blocks created by splitting the block. A block is a child of the top-level block (B0), an assembly block, or another block.

block decomposition

Breakdown of block units from the major parts to the piece part level. This process includes the complete assembly tree of a structural block that reaches to the lowest level structural part.

block division

Process of establishing boundaries of the major units or blocks to use in building a plant.

build strategy

Combination of block division, block decomposition, erection network, and assembly process that, when taken as a whole, represents the assembly plan and sequence for building a plant. Note that the plan does not include the schedule but does include the activities needed to develop the schedule.

built ships

Complete database of NGC information after completion of the ship contract.

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

ceiling

Overhead design of the cabin area, including distribution systems for power, water, and ventilation.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

class

Grouping of individual objects that share some very significant, common characteristics.

class rule check

Verification that the developing design meets the rules of a particular classification society, such as ABS, Lloyd's, or DNV.

Class Rules

Classification Society Design Rules.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

compartmentation

Division of the ship into functional spaces or compartments.

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

control point

A point that is used to control the shape of a NURBS curve or surface. Curves have a one-dimensional array of control points, while surfaces have a two-dimensional array.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

detail schedule

Lowest level of schedule used to manage and track work progress.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

erection clearance

The amount of space left between members to ease assembly.

erection network

Sequence of activities needed to erect block units.

external appendages

External structure attached to the hull, such as the propeller nozzle, shaft struts, bilge keel, and so forth.

fabricate

To cut, punch, and sub-assemble members in the shop.

face

A topological object that represents a trimmed surface bounded by a loop of edges.

face plate

An edge reinforcement type that places a plate or profile at the selected plate edge.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

fire integrity

Deck and bulkhead treatments and fire and smoke blocks for fire control and retardation.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

focus of rotation

A point or line about which an object or view turns.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

joiner

Non-structural bulkheads, and trim and built-in furnishings.

kinematics analysis

Analysis of mechanical motion.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

lifting

Analysis of the units that must be moved or positioned to determine weight and center of gravity (CG), unit height (including roll over requirements), and the appropriate number and location of lifting pads.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance parts

Required material for depot or on-board repair or overhaul of equipment, as determined by engineering study. Generally at a level below the purchased construction object of the plant.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

natural surface

A surface without a boundary curve.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the

case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

physical occurrence

Unique specific object that has traceability and is the physical manifestation of an occurrence object. A physical occurrence applies to one and only one hull. It is a version of its occurrence object with as-built or as-modified differences included and has a serial number or lot number.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

product

Data objects that describe the components of a ship and any corresponding properties. An individual object or part (or its representation in the product model) that may be installed in the ship. Examples of individual products include objects such as a coffee urn, a light fixture, a piece of pipe, a piece of ventilation duct, a radar display console, a bulkhead plate, and a structural profile stiffening a bulkhead.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

query select sets

Set of objects that are selected in a query or queries on the database.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

sheetbody

A topological object that represents a collection of faces joined along their common edges (stitched).

shell structure

External portion of the surface of the plant.

ship

A collection of modeled objects that can be simultaneously displayed and edited in a workspace. A Ship points to a Catalog (optionally shared with other Ships). Access control is managed at the Ship level.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

specials

An option category that allows you to control specialized calculations for equipment trim, repeatability, and center-of-gravity.

specifications

Contracted requirements for the plant.

steel outfitting

Internal structural elements of a ship that are required to meet a local requirement such as foundations, non-structural bulkheads, walkways, and so forth.

stern frame

Casting and structure that support the rudder and shaft opening.

stud

A bolt, threaded on both ends, used to connect components.

suspended floor

A concrete floor system built above and off the ground.

swash bulkhead

A longitudinal or transverse nontight bulkhead in a tank that decreases the swashing motion of the liquid contents. A plate in a tank that has this same effect but that does not extend to the bottom of the tank is called a swash plate.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

transverse

At right angles to the fore-and-aft center line.

transverse frames

The athwartship members that form the ribs of the ship.

trim

The difference between the forward draft and the aft draft.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

tumble home

The inboard slope of the side of a ship, usually above the designed waterline.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

vertical keel

A row of vertical plates extending along the center of the flat plate keel.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

water line

A line parallel with the base line that depicts the water.

watertight door

A door that when closed prevents the passage of water.

weather deck

A deck exposed to the weather.

weathertight door

A door that when closed prevents the passage of rain and spray.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

windlass

The machine used to hoist and lower anchors.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

Index

A

abaft • 211
abstract part • 211
Active Template Library (ATL) • 211
Add a custom batch process • 142
Add a seam • 40
aft • 211
after body • 211
aftermost • 211
angle • 211
Apply rule for production routing • 129
approval state • 211
arrangement (accommodation) • 211
assembly • 211
Assembly Arrangements • 154
assembly block • 212
Assembly Hierarchy • 15
assembly process • 212
Assembly Properties Dialog Box • 72
Assembly Sequence Form Dialog Box • 80
Assembly Views • 148
attribute • 212
Auto Assembly • 87
auto-nesting • 212
axis • 212

B

basic design • 212
Batch Process Selection Dialog Box • 144
bill of material (BOM) • 212
block • 212
Block Assignment • 156
block decomposition • 212
block division • 212
Block Properties Dialog Box • 51
build strategy • 212
built ships • 213
BuiltUpAutoAssemblyRule • 89
bulkload • 213

C

Candidate Tab (Edit Rule Settings Dialog Box) • 172
catalog • 213
Catalog database • 213
ceiling • 213
chain • 213
change history • 213
change management • 213
change propagation • 213
Check Manufacturability • 202
Check objects for manufacturability • 203
class • 213
class rule check • 213
Class Rules • 213
classification folder • 214
codelist • 214
commodity code • 214
commodity item • 214
Common Part Explorer • 166
Common Part Folder Properties Dialog Box • 169
Common Part Group • 132
Common Part Group Properties Dialog Box • 133
Common Part Manager Properties Dialog Box • 168
Common Part Service • 163
Common Part Utilities • 169
Common Parts • 163
Common Property Tabs • 32
Compare an object type • 170
Compare an XML • 171
Comparison Tab (Edit Rule Settings Dialog Box) • 172
compartmentation • 214
component • 214
concurrent access • 214
Configuration Tab • 32
consolidated tasks • 214
constraints • 214
contract • 214
control point • 214
coordinate • 214
coordinate system • 215
Create a common part group • 133
Create a new assembly or assembly block • 69
Create a new weld group folder • 96
Create a rule-based assembly • 88
Create block assignment log file • 160
Create common part managers • 131
Create Common Part Managers • 130
Create penetration spools • 117
Create Penetration Spools • 117
Create spools • 110

cutting plane • 215

D

damage records • 215
data interchange • 215
database • 215
database backup • 215
database break and recovery • 215
database copy • 215
database management • 215
database monitor record • 215
Define a coincident plane • 25
Define a plane using a point and a normal vector • 28
Define a plane using three points • 29
Define an offset plane • 26
Define plane by vectors normal • 29
Define plane using angle from plane • 27
Define the assembly sequence • 80
degree • 215
Delete a seam • 41
Delete a spool • 111
design alternative • 215
design approval log • 216
design data auto input • 216
design documents • 216
design object • 216
design progress check • 216
design review • 216
design service • 216
design standard • 216
detail schedule • 216
distributed systems • 216
distribution systems • 216
documentation • 216
drawing tool • 216

E

easting • 217
edge • 217
edge distance • 217
Edit a planning joint • 95
Edit an assembly or assembly block • 72
Edit Rule Settings Dialog Box • 171
Edit spool properties • 111
Edit the rule settings • 196
Editing Properties • 30
equipment catalog • 217
erection clearance • 217
erection network • 217
external appendages • 217

F

fabricate • 217
face • 217
face plate • 217
face-to-face • 217
fasteners • 217
feature • 217
fence • 217
field adjustment • 218
fire integrity • 218
Flatten an assembly or block • 210
Flatten Block/Assembly/Assembly Block • 210
flavor • 218
focus of rotation • 218
full penetration weld • 218
function points • 218
functional block diagram • 218
furnishings • 218

G

General Tab • 33, 134, 168, 169
General Tab (Assembly Properties Dialog Box) • 72
General Tab (Block Properties Dialog Box) • 52
General Tab (Planning Joint Properties Dialog Box) • 98
Generate a planning joint • 95
Generate Planning Joints • 90
Generate Production Routing • 127
Generate Spools • 110
Generic Batch Command Dialog Box • 143
generic specific • 218
GUIDs • 218

H

host location • 218
host server • 219

I

Icons in the Workspace Explorer • 17
Import Assembly Data from Excel • 161
Import Assembly Data from Excel Dialog Box • 162
initial design • 219
initial structural plan • 219
instantiation • 219
interference checking • 219

J

job order • 219
joiner • 219

K

kinematics analysis • 219
ksi • 219

L

leg length analysis • 219
library • 219
life cycle database • 219
lifting • 219
link • 220
lintel • 220
load group • 220
location • 220
Log Tab (Spool Generation Dialog Box) • 115
logical member • 220

M

machinery • 220
macro • 220
Main Tab (Seam Properties Dialog Box) • 146
maintenance envelope • 220
maintenance parts • 220
maintenance records • 220
Manage Block Intersections • 58
Manage Block Intersections Dialog Box • 62
Manage intersections • 61
Manufacturability Checking Settings Dialog Box • 204
Manufacturing Service Manager • 135
material analysis • 220
material list • 220
Merge • 77
Merge blocks or assemblies • 78
methods • 221
Mirror Assembly • 83
Mirror the assembly hierarchy • 85
Modify a block • 49
Modify a seam when leaf systems are not added or removed • 42
Modify Common Part Groups • 167
Modify the size of the top-level block (B0) • 50
Move a planning joint to a different assembly • 97
Move a planning joint to a different group folder • 96
move from point • 221
move to point • 221
MTO neutral file • 221

N

natural surface • 221
New (Block) Assembly • 64
node • 221
northing • 221
Notes Tab • 33
nozzle • 221
nozzle standout • 221
NPD (Nominal Piping Diameter) • 221

O

object • 221
occurrence (of part or equipment) • 221
occurrence property • 222
Occurrence Tab • 34, 120
Occurrence Tab (Spool Properties Dialog Box) • 116
Optional Schedule Properties Dialog Box • 161
Options Tab (Penetration Spool Generation Dialog Box) • 118
Options Tab (Spool Generation Dialog Box) • 112
origin • 222
origin point • 222
orthogonal • 222
orthographic • 222

P

P&ID • 222
package • 222
painting • 222
parameter • 222
part class • 222
part number • 222
PDS (Plant Design System) • 223
Penetration Spool Generation Dialog Box • 118
Penetration Spool Properties Dialog Box • 120
Permission Group Behaviors between Tasks • 35
physical occurrence • 223
PinPoint • 223
Plane Methods • 25

Planning • 10
Planning Joint Properties Dialog Box • 97
Planning Utilities • 197
Preface • 8
principle of superposition • 223
product • 223
Product Data Management (PDM) System • 223
product structure • 223
production planning • 223
promotion • 223

Q

query select sets • 223

R

Reevaluate parts for common part managers • 168
reference data • 224
Relationship Tab • 34
Remove Common Parts • 167
Repair Planning objects • 197
Repair Planning Objects • 197
Repair Planning Objects Dialog Box • 197
resource estimation • 224
Review a manufacturing object • 137
Review Dialog Box • 138
Review or update a set of objects selected by a filter • 137
route • 224
Rule Variables (Edit Rule Settings Dialog Box) • 195
Run a custom batch process • 142
Run a detailing batch process • 141
Run a production batch process • 141
Run Report • 147

S

satellite server • 224
Schedule [Task] Dialog Box • 160
Schedule a planning joint using Batch Services • 97
Schedule block assignment using Batch Services • 159
Schedule check manufacturability using Batch Services • 203
schema • 224
schema update utility • 224
Seam Properties Dialog Box • 145
Select a common part group using filters • 133
Select or Modify Frames Dialog Box • 56
Selection of Possible Report Data Dialog Box • 139
Sequence • 79
Set assembly manufacturing orientation by object • 71
Set assembly user-defined planar manufacturing orientation • 70
Set block manufacturing orientation by baseplate • 50
Set block manufacturing orientation by three-point plane • 51
sheetbody • 224
shell structure • 224
ship • 224
Show Assembly Sequence • 150
Show Assembly View • 149
Show Manufacturability Inconsistencies Dialog Box • 205
Show parts • 149
Show Parts • 148
Show Selected • 148
Show selected objects • 148
Show the assembly view • 150
site • 224
site administrator • 224
site setup • 224
sketch and trace • 225
SMAutoAssemblyRule • 88
specials • 225
specifications • 225
Split a block • 49
Split a planning joint • 107
Split Block • 43
Split Notification • 39
Split Planning Joint • 106
Spool Generation Dialog Box • 112
Spool Properties Dialog Box • 116
Spooling • 108
Spooling Tab • 112, 118
Start Common Part Service • 165
Start the block assignment process • 158
steel outfitting • 225
stern frame • 225
Stop the block assignment process • 159
stud • 225
Submit Batch Job • 140
suspended floor • 225
swash bulkhead • 225
symmetric node • 225
Synchronize Hierarchy • 154
Synchronize the assembly hierarchy • 155
system • 225

T

tag number • 225
target point • 225
Test block assignment • 200
Test Block Assignment • 200
Test Block Assignment Dialog Box • 200
Test Rules • 169
tolerant geometry • 226
transverse • 226
transverse frames • 226
trim • 226
trimmed surface • 226
trunk • 226
tumble home • 226

U

unit/module modeler • 226
Un-Split • 145
Un-split a plate at a selected seam • 145
Update a common part folder • 168
Update a common part manager • 168
Update a manufacturing object • 137
Use a drag-and-drop operation to assign parts • 62
user attributes • 226

V

version control • 226
vertex • 226
vertical keel • 226
View Common Parts • 167
View the assembly sequence • 153
viewset • 227

W

water line • 227
watertight door • 227
weather deck • 227
weathertight door • 227
weight and CG analysis • 227
welding • 227
What's New in Planning • 8
windlass • 227
wirebody • 227
wizard • 227
work content • 227
work order • 227
working plane • 227
workset • 228
workspace • 228
workspace document • 228
Workspace Explorer • 228